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Preface

This document is a user's guide for the Intergraph Smart[™] 3D Drawings and Reports task and provides command reference information and procedural instructions.

The Drawings and Reports user documentation is delivered in three separate documents:

- Orthographic Drawings User's Guide
- Isogen Isometric Drawings User's Guide
- Reports User's Guide

Documentation Comments

For the latest support information for this product, comments or suggestions about this documentation, and documentation updates for supported software versions, please visit *Intergraph Smart Support (https://smartsupport.intergraph.com)*.

What's New in Drawings and Reports

The following changes have been made to the Drawing and Reports task.

Version 2016 (11.0)

Drawings (General)

- You can now transfer the ownership of drawings between permission groups. For more information, see Configuration Tab (Properties Dialog Box) (on page 68). (P1 CP:278233)
- A new 3D Dimension ribbon in SmartSketch Drawing Editor overrides manual 2D dimensions with 3D dimensions. For more information, see *Dimensioning in 3D Drawings* (on page 267) and 3D Dimension Ribbon. (P1 CP:139408)
- You can copy a drawing view only by using the Copy and Paste View command. For more information, see Copy and Paste View Command (on page 375). (P2 CP:226825)
- You can perform **Update Selected** on both full and cropped views. For more information, see *Drawings View Explorer* (on page 256). (P2 CP:230723)
- Added a new ribbon, Generic View Ribbon, to the Associate Objects to View command that allows you to specify options on a selected graphic view with a generic rule set view style. For more information, see Associate Objects to View Command (on page 300). (P2 CP:244388)
- You can now customize the shortcut menu on a folder in the Management Console. For more information, see New Command (on page 41). (P2 CP:255695)
- You can now either delete a manual view placed by rule, or move the view to the UnAssigned Folder. For more information, see *Delete Views* (on page 415). (P2 CP:256645)
- You can now customize Drawings by Rule Steel Order rulesets in .NET to generate drawings. For more information, see *Drawings by Rule Components* (on page 92). (P2 CP:266707)

- In drawings by rule, view annotations, such as the ruler and view name, are automatically adjusted in relation to the **View Offset** value. For more information, see *View Tab (Drawing View Properties Dialog Box Drawings View Explorer)* (on page 283). (P2 CP:267889)
- Smart 3D displays a warning telling you to use **Copy and Paste View** in rather than the traditional **Cut**, **Copy**, and **Paste** commands with views. For more information, see *Copy and Paste View Command* (on page 375). (P2 CP:226825, CP:275427, and CP:284427)
- You can now prevent landing curves from occluding other landing curves using the AddSolidForVHL flag. For more information, see Generic Stiffener Landing Curve. (P2 CP:296036)
- Group Selected Labels stacks selected objects without overlapping the leader lines. For more information, see *Group Selected Labels* (on page 339). (P2 CP:265171 and P3 CP:271343)
- You can select multiple objects with the Place a Label command, and place a group label for the objects. For more information, see Place a Label Command (on page 322) and Group Labels (on page 326). (P3 CP:181864)
- Drawing by Rule view properties only display applicable drawing by rule view styles. For more information, see View Tab (Drawing View Properties Dialog Box - Drawings View Explorer) (on page 283). (P3 CP:215229)
- You can place a manual view for non-shell plates in Marine mode. For more information, see Place a Manual View for Non-Shell Plates (on page 292).
- You can now update all drawings by rule documents with the Full Update action in Intergraph Smart Batch Services. For more information, see Full Update Action. (P1 CP:160700) (P3 CP:199390)
- A new property, View Cone Angle, displays the cone angle value. For more information, see View Tab (Drawing View Properties Dialog Box Drawings View Explorer) (on page 283). (P3 CP:266696)
- Added a new Warning view state to the Drawings View Explorer. For more information, see Drawings View Explorer (on page 256). (P3 CP:271093)
- In ruleset views, if you change the view style before editing View Offset for the first time,
 View Offset displays the offset value as defined in the .xml template of the view style. For more information, see View Tab (Drawing View Properties Dialog Box) (on page 277). (P3 CP:272849)
- A new file option, File Already Exists Action, specifies how to save the hierarchy if a file
 with the same name already exists. For more information, see Save As Command (on page
 70) (P4 CP:273965)
- Any time you update a 3D model document, the software now generates a single log file containing status information and any errors encountered during the process. The log file location is %TEMP%\EFUpdateCache\[3D Model ID]\[3D Model ID]\[3D Model ID]\]. (P2 CP:292733)
- Added information about hiding plant view styles when working in marine mode. For more information, see View Tab (Drawing View Properties Dialog Box - Drawings View Explorer) (on page 283). (P3 CP:293501)

Orthographic Drawings

- You can now export 3D model graphics directly to MicroStation V8 DGN file format. For more information, see Exporting 3D Model Graphics to MicroStation (on page 194). (P4 CP:251155)
- Added a new MicroStation Version option to Setup dialog box (MicroStation DGN Files).
 For more information, see Setup Dialog Box (MicroStation DGN Files) (on page 194). (P4 CP:112450)
- Added information about the working units supported by the seed file when exporting graphics to MicroStation V7 DGN file format. For more information, see *Create MicroStation* DGN files (on page 195) .(P2 CP:75204)
- Highlight Annotations and Clear Manual Edits commands are available on the Drawings Compose toolbar. For more information, see *Highlight Annotations Command* (on page 376) and Clear Manual Edits Command (on page 379). (P2 CP:256406)
- You can now associate manually placed objects to graphic views. For more information, see Associate Graphics to Graphic View Command (on page 379). (P2 CP:247816)

Isogen Isometric Drawings

- You can now produce cable tray and HVAC isometric drawings. To support this enhancement, the Style list on the Setup dialog box now enables you to filter styles according to the isometric drawing type. For more information, see Setup Dialog Box (Isogen Isometric Drawing by Query Component).(P2 CP:273785 and CP:209035)
 - ★ IMPORTANT In this version of the software, cable tray and HVAC isometric drawing options are limited to beta access. As such, isometric options specific to cable tray and HVAC drawings are not visible in **Isogen Configuration** by default. To turn on cable tray and HVAC options so that you have access to the full array of isometric drawing settings provided by the software, please contact *Intergraph Support* (http://www.intergraph.com/support).
- To support the creation of cable tray and HVAC isometric drawings, two new drawing styles are delivered with the software: Iso_CableTray and Iso_HVAC. For more information, see Isometric Drawing Styles.
- The function of the piping component file (PCF) has been extended. In addition to piping content, the PCF now supports the transfer of HVAC and cable tray content and configuration information between Smart 3D and Isogen.
- You can now associate manually placed objects to graphic views. For more information, see Associate Graphics to Graphic View Command (on page 379). (P2 CP:247816)
- In previous versions of the software, you used TreatTapsOnPipeAsTeeStubs to specify whether the software managed taps as tee stubs or as regular taps when generating a piping isometric drawing or when creating a piping component file (PCF). The software now treats all radial taps placed on pipe as set-on tees, or tee stubs. All non-radial taps remain as taps. (P2 CP:255192)
- Previous versions of Smart 3D refer to Piping Isometric Drawings. Because the software now supports the production of cable tray and HVAC isometric drawings, the UI now reflects the generic term Isogen Isometric Drawings. This document has been updated to reflect this change in the UI. (P2 CP:259784)

- The Isometric Style Options Browser, which enabled you to control all of the options related to the appearance and information content of the isometric drawing, has been replaced with Isogen Configuration. For more information, see Isogen Configuration. (P2 CP:255749)
- You can now show the slope symbol without any angle/ratio for sloped lines. (P3 CP:157751)
- Isogen Configuration supports the ability to import a style fragment. For more information, see Import data from a style XML file. (P2 AL:14940)
- You can export a customized set of isometric drawing properties, Alternative Text, and report definitions to a style .xml file. For more information, see Export an isometric drawing style fragment and Export Style Fragment Dialog Box. (P2 AL:14965)
- Isogen Configuration supports a new file path macro, \$SYMBOLSHARE\$. This macro enables you to reference the SharedContent folder when specifying any path within an isometric drawing style. Previous versions of the software required you to specify a hard-coded path to reference any file located in the SharedContent folder. (P2 CP:105056)
- PCF syntax now supports the following material attributes: GEOMETRIC-STANDARD, MATERIAL-OF-CONSTRUCTION, OUTSIDE-DIAMETER, OUTSIDE-DIAMETER2, RATING, RATING2, SCHEDULE, SCHEDULE2, WALL-THICKNESS, and WALL-THICKNESS2. (P2 CP:248316)
- A new PDF document is delivered with the software. The Isometric Drawing Options Reference Guide contains descriptions and general information for working with the Isogen and Smart 3D isometric drawing properties. In previous versions of the software, this content was delivered as part of the Piping Isometric Drawings User's Guide. The new document (IsometricDrawingOptionsReferenceGuide.pdf) is available in the [Product Folder]:\Documentation\Help folder and from the Help > Printable Guides command in the software.
- Added guidelines for creating a dotted dimensioned label. For more information, see Create a dotted dimensioned label.
- The following changes are specific to the isometric drawing properties supported by the software. For more information, see What's New in the Isometric Drawing Options Reference Guide. You can access this document using the Help > Printable Guides command in the software.
 - A new options group, Grouping, has been added to Drawing Area > Graphics. (P2 AL:10443)
 - The Group graphics by UCI property allows you to control how graphics are grouped in the output CAD file. (P2 AL:10443)
 - The Welds and Joints style category has been re-organized to improve its overall layout. (P2 AL:14803)
 - The software now supports horizontal positioning of the dimension standout. (P2 AL:11970)
 - The software supports the ability to pull properties directly from the Drawing Sheet object. (P2 CP:123112)
 - The number of user-definable Alternative Text (AText) strings that the software supports has been expanded. (P2 CP:74293)

- A new OidType called Drawing Sheet enables you to pull properties directly from the Drawing Sheet object.(P2 CP:123112)
- The software supports output of a double-banked material list on the isometric drawing. Previous versions of the software supported offsetting the material list section only along the Y (or vertical) direction of the drawing using Section2 Y Offset and Section3 Y Offset. Now you can also offset the material list section along the X (or horizontal) direction of the drawing using Section2 X Offset and Section3 X Offset. You can set this option using Materials > Drawing Material List > User Defined. (P3 CP:250733)
 - **NOTE** You can also use the **Drawing Setup Tool** to set the horizontal offset for the report data contained in Section 2 and Section 3 of the material list. For more information, see **X** under **Section Two** and **Section Three** in Drawing Areas Page and Material List Page.
- The software supports user-specified reference dimension placement locations on a piping isometric drawing. As part of this new functionality, the **Placement** property is renamed to **Automatic Placement**, and a new setting, **Off**, has been added. (P3 CP:176166)
- Added information about using pipeline data text attributes to dynamically name Isogen output files.

Reports

- You can now specify a unit delimiter between primary and secondary units in a report. For more information, see Parameters (Report Shortcut Menu). (P2 CP:140982)
- Added a new report, Designed Member Itemized Material Take-off. For more information, see Designed Member Itemized Material Take-off. (P2 CP:234498)
- Added the trench run report. For more information, see Trench Run Report (Civil). (P2 CP:259491)
- Added a new report, Diagnostic Synch Workspace Report (Diagnostic). For more information, see Diagnostic Synch Workspace Report (Diagnostic). (P2 CP:269935)
- Added information on using the Reporting Requirement property to exclude objects from reports. For more information, see Report Queries to Extract Data. (P3 CP:85240)
- Added the workflow for configuring and running personal reports. For more information, see Run Report. (P4 CP:281013)

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SECTION 1

Drawings and Reports

The Intergraph Smart[™] 3D Drawings and Reports task creates orthographic drawings (by a variety of methods), isometric drawings, and reports from the model. When the 3D model changes, you can update your drawings and reports.

The Drawings and Reports task is also responsible for publishing your drawings and reports. When your model is registered using the SmartPlant Registration Wizard, you can publish volume and composed drawings, orthographic drawings, isometric drawings, and reports. You can also publish 3D model data using the 3D Model Data component.

The **Management Console** organizes the different document types into a customizable hierarchy. Using the component functionality of the console, you can create, edit, update, print, save, and publish the deliverables. This hierarchy of components and documents is also available in many of the 3D tasks, such as Common, by using the **Tools > Drawing Console** command. You can perform nearly all of your document operations using this command. For more information, see the *Common User's Guide*.

Before you can create components for drawings and reports, your administrator must organize the **Management Console** hierarchy with folders for each component type. Then, the administrator must complete several setup steps, including setting up drawing and report templates, creating view styles, creating appropriate filters, and specifying isometric drawing options. Default templates and view styles are delivered with the software, and you can customize them to suit your needs.

It is possible to customize templates and view styles before any objects exist in the model. However, to create drawings and reports, objects must exist in your model. For example, if you want to generate Isogen isometric drawings, you must have piping, cable tray, or HVAC data in your model.

Composed Drawings

Composed drawings are orthographic drawings created in a 3D task such as Common. The composed drawing component, available in the Drawing and Reports task **Management Console**, manages the composed drawings you create. Composed drawings are flexible, allowing you to have views that are managed by a drawing region and associate the views to volumes and other views.

Volume Drawings

Volume drawings are useful for creating general arrangement or construction drawings of areas within the model. In the Volume Drawing workflows, you or your administrator must create or edit border templates. You can place drawing property labels in the title block of the template to fit your company or project. You also must configure the view styles, which are sets of rules that determine how the graphics in the three-dimensional model are represented on the drawings. View styles use filters. You can create a folder of drawing filters, with new, existing, and future filters for each discipline. You place drawing volumes in the Space Management task. You can publish Volume drawings when they are up-to-date.

Orthographic Drawings by Query

The Orthographic Drawing by Query component, in conjunction with the Drawings by Query Manager component, creates drawings in mass by specifying a filter-based query to collect objects for drawings. This drawing type is appropriate for creating detail drawings of particular objects within the model. They are especially useful when creating drawings that use the same style or format for large numbers of similar objects, such as hangers or supports.

Just like composed and volume drawings, you can print, update, save into MicroStation J (V7 and V8) or AutoCAD formats, or publish Orthographic Drawings by Query. When you publish Orthographic Drawings, a viewable graphic file is created; no physical data is published.

Orthographic Drawings by Rule (marine mode and material handling mode)

Drawing by Rule types include design and detail drawings, manufacturing drawings, and assembly drawings. You can use the Drawings by Rule component to create custom drawings, or select a delivered Drawings by Rule package to create your drawings.

Isogen Isometric Drawings by Query

Like Orthographic Drawings by Query, you create Isogen Isometric Drawings by Query by specifying a filter-based query to collect the objects. The workflow requires that you create or edit border templates to fit your company or project. You or your administrator also must set the isometric options for each of the isometric styles that you need in your project.

When you publish Isogen Isometric Drawings by Query, they are published as viewable graphics. No physical data is published.

Reports

In the Spreadsheet Reports workflow, you create report templates, which control the content and format of reports. The default file format of reports in the software is Microsoft Excel® format. The Report Template Editor provides the ability to configure your reports to use queries and special formatting.

You can publish Spreadsheet Reports just like drawings. However, the Spreadsheet Reports are published as Excel spreadsheets; no physical data is published.

■ NOTES

- Microsoft Excel 2003 is the minimum supported version for the Drawings and Reports task.
- Microsoft Excel 2007 has a file format and extension of .xlsx. However, the delivered report templates still use the old .xls extension. If you create a report template with the .xlsx extension, only machines with Excel 2007 will be able to handle (edit, update, or open) the report. If you attempt to open an .xlsx file with an older version of Excel, an error message displays stating that the file is not compatible with the version of Excel.
- If you use Office 2003, in Microsoft Excel under Tools > Macro > Security > Trusted Publishers tab, check the Trust Access to Visual Basic Project option.
- If you use Office 2007 and Office 2010, click the Microsoft Office button to open Excel Option. Go to the Trust Center category and select the Trust Center Settings button. Select the Macro Settings category and check Trust access to the VBA project object model.
- For more information about Microsoft Office and service packs, refer to the *Microsoft web site* (http://www.microsoft.com/) (http://www.microsoft.com/).

ISO Standards in Marine Drawings

Intergraph Smart $^{\text{TM}}$ 3D marine drawings are ISO compliant. Several ISO templates are delivered with the product.

Below is a list of ISO Reference numbers and the description of the document.

ISO 128-20:1996(E)	Technical Drawings – General principles of presentation Basic conventions for lines
ISO 128-21:1997(E)	Technical Drawings – General principles of presentation Preparation of lines by CAD systems
ISO 128-22:1999(E)	Technical Drawings – General principles of presentation Basic conventions for applications for leader lines and reference lines
ISO 128-23:1999(E)	Technical Drawings – General principles of presentation Lines on construction drawings
ISO 128-24:1999(E)	Technical Drawings General principles of presentation Lines on mechanical engineering drawings
ISO 128-25:1999(E)	Technical Drawings General principles of presentation – Part 25: Lines on shipbuilding drawings
ISO 128-30:2001(E)	Technical Drawings – General principles of presentation – Part 30: Basic conventions for views
ISO 128-40:2001(E)	Technical Drawings – General principles of presentation – Part 40: Basic conventions for cuts and sections
ISO 128-50:2001(E)	Technical Drawings – General principles of presentation – Part 50: Basic conventions for representing areas on cuts and sections
ISO 5457:1999(E)	Technical product documentation – Sizes and layout of drawing sheets
ISO 7200:2400(E)	Technical product documentation – Data fields in title blocks and document headers
ISO 5261:1995(E)	Technical Drawings – Simplified representation of bars and profile sections
ISO 2553:1992(E)	Welded, brazed and soldered joints – Symbolic representation on drawings
ISO 5572:1987(E)	Shipbuilding and marine structures – Numbering of equipment and structural elements in ships
ISO 7462:1985(E)	Shipbuilding – Principal ship dimensions. Terminology and definitions for computer applications
ISO 8193:1984(E)	Shipbuilding – Shell plating information

ISO 9203-1:1989(E)	Shipbuilding – Topology of ship hull structure elements – Part 1: Location of elements
ISO 9203-2:1989(E)	Shipbuilding – Topology of ship hull structure elements – Part 2: Description of elements
ISO 9203-3:1989(E)	Shipbuilding – Topology of ship hull structure elements – Part 3: Relations of elements
ISO 129-1:2004(E)	Technical drawings – Indication of dimensions and tolerances – Part 1: General principles
ISO 6428:1982(E)	Requirements for microcopying
ISO 3898:1997(E)	Bases for design of structure – Notations – General symbols
ISO 3098-0:1997(E)	Technical product documentation – Lettering – Part 0: General requirements
ISO 3098-2:2000(E)	Technical product documentation – Lettering – Part 2: Latin alphabet, numerals and marks
ISO 3098-5:1997(E)	Technical product documentation – Lettering – Part 5: CAD lettering of Latin alphabet, numerals and marks

By default, the following metric drawing templates are delivered in the ISO format.

LARGE

- Paper Format (ISO 5457)
 - A0 (valid for A1 and enlarged formats)
 - Title-block (ISO 7200)
- Lines (ISO 128 Part.25, ISO 6428)
 - Wide 0.7 mm
 - Narrow 0.35 mm
- Characters (ISO 3098-5, ISO 6428)
 - Titles 7 mm (ISO 3098 BVL 7)
 - Labels 3.5, bold or 5 (ISO 128 Part.22)
 - Dimensions 3.5, values rounded to 1 mm (ISO 129)
- Symbols to match above lines and characters for
 - Relative positions of structural elements (BL, CL,...)
 - Sections and details (Section plane, Section and Detail Titles)
 - Profile cross-sections (ISO 5261)
 - Profile end-cuts re-symbolization

- Section and plates butt joints
- Welds (ISO 2553)

SECTION 2

Delivered Drawing Types

The delivered drawing types are drawing templates that include drawing borders, documentation annotation, note areas, and selection and resymbolization criteria.

Several drawing types are delivered and fully designed to meet particular drawing requirements. You can use the delivered types to create new drawing types and to modify the view styles or border templates as needed. You can copy a template from an existing drawing or you can copy volumes only, allowing you to create multiple drawings with the same border graphics. To copy a drawing type component, select the item on the **Management Console** hierarchy, and then select **Copy**. To paste the item, right-click a location in the hierarchy or in the **Detail View** and select **Paste**.

All of the delivered drawing types provide customizable templates and view styles. The delivered **Equipment Plan** drawing type is provided as an example below:

- The **Equipment Plan** is a single view drawing plan. It includes the location of equipment, structural columns, building walls, equipment steel, vessel and mechanical steel, and roads.
- The Equipment Plan includes general information for coordinate systems, sheet scales, and modifications. The drawing border provides the border graphics, title block graphics, and the title block labels.

The document annotation includes the following:

- North Arrow Indicates the orientation of the model. The large symbol is used, which is typical for single view drawings. The north arrow is placed on each drawing view. Click and drag the symbol to position it within the Note Area if required.
- Key Plan Shows the geographic position of the single grid relative to the rest of the grids
 of the same type in the single block.
- Drawing Notes Shows the collection of notes, which consists of general notes applicable
 to all drawings, notes applicable to a discipline, notes applicable to a category of drawings,
 and notes specific to a single drawing (such as an element or a border report).

Note Area

The Note Area is used to display drawing notes and key plans. The note area on the Equipment Plan is five inches on the right-hand side of the drawing border. This area extends from the top of the border down to the top of the title area border. The Note Area is optional, and is not a required element in the template.

View Regions

The View Region defines the drawing view arrangements. The **Equipment Plan** defaults to a single view with a five-inch Note Area and one-inch margins around the drawing view.

Drawing View

The **Equipment Plan** is a single view plan. The following specifications are set:

Direction - Set to Plan view (Looking Plan).

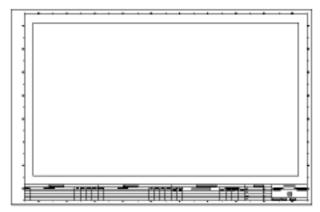
- Rotation Depends on the volume type and the Coordinate System defined, based on the grid section's X and Y size. For example, a volume placed by four points in the Space Management task will rotate the view.
- Scale Set to 1/4" = 1' or 1 = 50.
- North Arrow One symbol included per drawing view. Click and drag the symbol to the Note Area.
- Annotation The plan uses matchline labels to indicate the appropriate coordinate of that segment of the grid boundary along with continuations. The matchlines are lines that outline the boundaries of a grid by following the exterior boundaries of the collection of sectioning elements defining the grid represented in the view.
- Drawing Volumes The drawing volume is the queried 3D volume in the model.

View Styles

The view style specifies the object filters included in the drawing if present in the queried 3D volume. It specifies how objects are displayed, including graphical representation, labeling, and dimensioning. In the **Equipment Plan**, the volume and composed drawing view style definitions are the same.

Title Blocks

The title block generally displays at the bottom of a drawing template. It can include signatures, revision and issue information, and other properties associated with the drawing.



You can add drawing properties to the title block using the **Place Drawing Property Label** command when editing a template.

See Also

Place a Drawing Property Label on a Template (on page 393) Drawings and Reports Naming Rules (on page 95)

Drawings by Rule Types

The delivered marine mode drawings by rule components provide easy starting points for more complex drawings. You can also use the generic Drawings by Rule component to create custom drawings.

Drawings by Rule

- Assembly drawings
- Hull Lines drawings
- Scantling drawings
- Expansion drawings
- Manufacturing drawings
- Offshore drawings

For information more information, see *Drawings by Rule* (on page 102).

Other Drawing Types

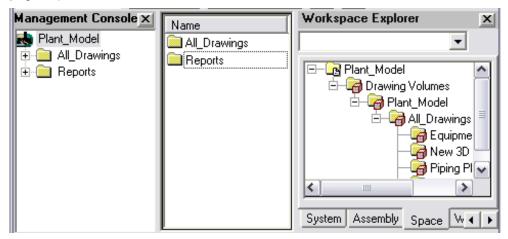
You can also use the following drawing types to create outfitting drawings:

- Composed and Volume drawings See Composed Drawings (on page 201) and Volume Drawings (on page 225).
- Orthographic Drawings by Query See Orthographic Drawings by Query (on page 213).
- Isogen Isometric Drawings by Query See Isogen Isometric Drawings by Query in the Isogen Isometric Drawings User's Guide.

SECTION 3

Interface Overview

This task includes different windows or views within its interface. You can toggle the display of these windows using commands on the **View** menu. For more information, see *View Menu* (on page 28).



The **Management Console** contains a hierarchy of folders and components that you create. If you right-click an item in the **Management Console**, the available menu commands vary, depending on the item and your permissions. For more information on managing folders and components in the **Management Console**, see *Shortcut Menus* (on page 33). The **Management Console** only displays in the Drawings task.

The **Detail View** shows the children of the selected item in the **Management Console**. You can select multiple components or documents by pressing **Ctrl** or **Shift** while selecting. To specify the columns in the **Detail View**, right-click a column heading and click **More**. The **Detail View** is overlaid by other windows depending on the current operation. For example, when reviewing drawings using the **Open** command, a 2D viewer displays. When you edit report templates, a tabular editor displays. For more information on setting the appearance of the Detail View, see *Detail View (View Menu)* (on page 29).

In plant mode, the **Workspace Explorer** is the tabbed view of systems, assemblies, spaces, and Work Breakdown Structure (WBS) items in the software. For more information about the **Workspace Explorer**, see the *Common User's Guide* available from the **Help > Printable Guides** command in the software. The **Workspace Explorer** does not display in the Drawings task.

■ NOTE Another window you use while working in this task is the SmartSketch Drawing Editor window, which displays as a separate application window. It allows you to edit border templates, drawing templates, and backing sheets for all types of drawings.

See Also

Menus and Toolbars (on page 25) Icons for Components and Drawings (on page 25) Permissions Overview (on page 27)

Menus and Toolbars

In this task, the commands available change according to the active window, selected **Management Console** or **Detail View** item, and the specific workflow.

For example, when you edit a drawing template or open a drawing, you control the template with **SmartSketch Drawing Editor**. You use **SmartSketch Drawing Editor** menus and toolbars to edit the open drawing template. When you edit report templates, the report menus and commands are available.

In addition, the shortcut menu that displays when you right-click an item in the **Management Console** or in the **Detail View** differs according to the type of item. For example, some of the commands on the shortcut menu for an Isogen isometric drawing are different from the commands on the shortcut menu for a composed drawing component.

For the root node of the **Management Console** hierarchy and Folder items that have no child items beneath them, the shortcut menu includes **Batch...**, **Copy**, **Paste**, **Delete**, **Rename**, **New...**, **Save Package...**, and **Properties**. If there are any child items beneath the root node or beneath the selected folder, the following commands are added: **Create Drawing(s)**, **Refresh**, **Run Query**, **Update Now**, **Print**, and **Save As**. If you are registered with SmartPlant Foundation, the **Publish** and **Revise** commands are added as well.

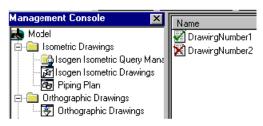
Also, keep in mind that the main menu bar available in the application varies by task. Some commands available in other tasks may not be available in this one.

See Also

Interface Overview (on page 24)
Detail View (View Menu) (on page 29)

Icons for Components and Drawings

The **Management Console** and **Detail View** display different icons to show the type and status of packages, components, and documents.



Drawing Type Icons

- Root of the model hierarchy
- Folder
- Search folder
- Imported folder
- 👊 Query manager
- 💁 Generic module folder

- Drawing by Rule component (marine mode and material handling mode only)
 Volume Drawing component
 Composed Drawing component
 Orthographic Drawing component
- Isogen Isometric Drawing component
- Spreadsheet Report component
- 3D Model Data component
- MicroStation 3D DGN drawing component
- Package of drawing components (in the **Add Component** dialog box)
- Drawings by Rule package (in the **Add Component** dialog box) (marine mode and material handling mode only)
- Drawing document. A status icon is always superimposed over this icon.

Document Status Icons

These icons are superimposed on the document icon and indicate document status.

- Submitted or scheduled for batch processing. These documents are either submitted or scheduled for batch processing.
- Updating or publishing. A document also displays this icon if an error occurred and forced the machine to end the update process before it completed. Right-click the drawing document, and select **View Log** for more information, or update the document again.
- ➤ Out-of-date. This document has been altered in SmartSketch Drawing Editor or the drawing properties have been changed. A document is not marked out-of-date if the 3D model has been changed. For example, changing the view style of a drawing view causes the document to be out-of-date, while moving a pump in the 3D model does not affect the drawing status (unless a Refresh is performed).
- Up-to-date. This document is an accurate representation of the 3D model based on the last update performed. If an object in the 3D model that is included in the drawing view has been moved inside the 3D environment, the document is still up-to-date unless **Refresh** is performed. A change must be made to the drawing properties or inside the drawing in order for it to be marked out-of-date without a **Refresh**.
- Error status. An error has occurred during the update process. Right-click the drawing, and select **View Log** for more information.
- S No graphic objects in the model associated with this drawing document. For example, the drawing is a piping isometric drawing document created from a Pipeline System that has no piping parts associated with it.
- The drawing document is a version 6.1 legacy Snapshot drawing. You should use the **Tools > Convert Legacy Snapshots** command to convert this document to a Composed Drawing for use in the current version of the software. If you do not convert the legacy snapshot

drawing, you cannot perform edit operations on the drawing, including update, revise, and publish.

See Also

Components Overview (on page 89)
Interface Overview (on page 24)
Menus and Toolbars (on page 25)
Update Now (on page 80)
Convert Legacy Snapshots (on page 240)

Permissions Overview

Your site administrator sets permissions and creates permission groups in the Project Management task. These permissions are used in the different tasks in the software to control user access.

You can see your current permission group in the dropdown box in the upper left-hand corner of the window when in the Drawings and Reports task.



The permission group to which an item belongs can affect the actions allowed against that item. For example, the propagation of properties down the hierarchy, from parent to child, is interrupted when a node or document in a read-only permission group is encountered.

The following list shows the actions relating to drawings and reports that are affected by permission groups:

- Accessing shortcut menu commands in the Management Console and Detail View
- Creating items, such as drawings, drawing views, and drawing volumes
- Propagating properties down through the hierarchy
- Deleting items
- Updating items, such as re-extracting drawings

In addition, access to the SharedContent share on the server computer affects actions such as creating and editing view styles and graphic rules.

See Also

Interface Overview (on page 24)
Menus and Toolbars (on page 25)

SECTION 4

View Menu

In This Section

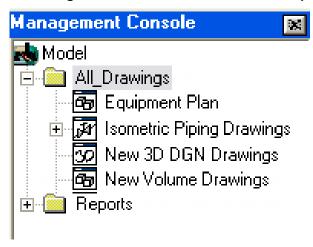
Management Console (View Menu)	28
Detail View (View Menu)	
Workspace Explorer (View Menu)	
Refresh (View Menu)	

Management Console (View Menu)

Toggles the display of the **Management Console**. By default, the **Management Console** is visible when you enter this task.

The **Management Console** contains a hierarchy of folders and components that you create. If you right-click an item in the **Management Console**, the available menu commands vary, depending on the item and your permissions. For more information on managing folders and components in the **Management Console**, see *Shortcut Menus* (on page 33).

■ NOTE When you switch to a different task and then return to the Drawings and Reports task, the Management Console remembers the node you last selected on the console hierarchy.



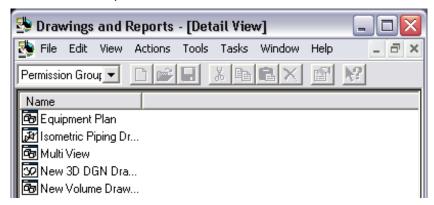
See Also

Interface Overview (on page 24)
Detail View (View Menu) (on page 29)
Workspace Explorer (View Menu) (on page 32)
Refresh (View Menu) (on page 32)

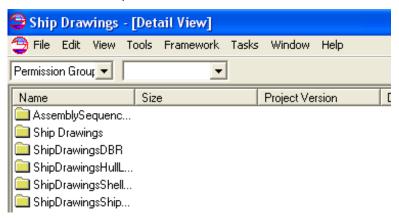
Detail View (View Menu)

Turns the display of the **Detail View** on and off. This command is located on the **View** menu. When checked, the **Detail View** is visible in the application window. When you right-click folders or documents in the **Detail View**, shortcut menus display. The items on the shortcut menu vary depending on the selected item, as shown in the following examples. For more information on the commands, see *Shortcut Menus* (on page 33).

Plant mode example:



Marine mode example:



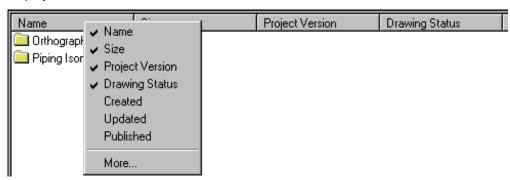
To modify the headings in the **Detail View**, right-click the column-heading area. Select **More** on the shortcut menu to display the **Column Settings** dialog box.

See Also

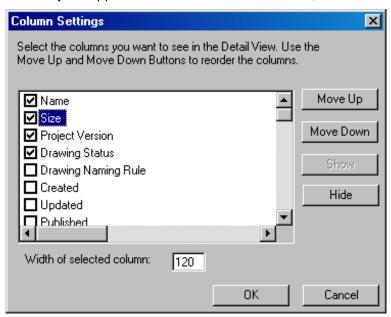
Specify columns in the detail view (on page 30) Column Settings Dialog Box (on page 31)

Specify columns in the detail view

1. Right-click a column heading in the **Detail View**. The shortcut menu shows the currently displayed columns with a checkmark ✓.



- Add and remove columns automatically by checking and unchecking them on the shortcut menu.
- 3. To modify the appearance and order of the columns, click **More** on the shortcut menu.



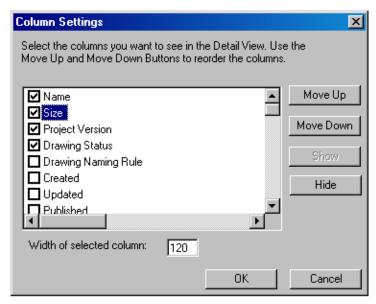
- 4. On the **Column Settings** dialog box, select the columns you want to include in the **Detail View**. Clear, or uncheck, the ones you do not want to include. You can also use the **Show** and **Hide** buttons to add and remove columns.
- 5. To change the order of the columns, click **Move Up** and **Move Down** on the **Column Settings** dialog box.
- 6. Specify the width of a column by selecting it and typing an integer in the **Width of selected column** box. You can also resize columns by dragging the edge of the column in the **Detail View**.

See Also

Interface Overview (on page 24)
Management Console (View Menu) (on page 28)
Refresh (View Menu) (on page 32)
Menus and Toolbars (on page 25)
Column Settings Dialog Box (on page 31)

Column Settings Dialog Box

Specifies the columns you want to see in the **Detail View**. You also can specify the order and width of the columns. You access this dialog box when you right-click in the column heading area of the Detail View and select **More** on the shortcut menu.



Move Up

Moves the selected column up one position. The column displays one position to the left in the **Detail View**.

Move Down

Moves the selected column down one position. The column displays one position to the right in the **Detail View**.

Show

Displays the column in the **Detail View**.

Hide

Hides the column in the Detail View.

▶ NOTE In addition to the **Show** and **Hide** commands, you can use the checkboxes beside the column names to add and remove them from the **Detail View**. Checked indicates that the column displays in the **Detail View**.

Width of selected column

Specifies the width of the column in pixels. You can specify a different column width for each column.

See Also

Interface Overview (on page 24)
Detail View (View Menu) (on page 29)
Management Console (View Menu) (on page 28)
Refresh (View Menu) (on page 32)
Menus and Toolbars (on page 25)

Workspace Explorer (View Menu)

Toggles the display of the **Workspace Explorer** as a viewer only in the Drawings and Reports task. By default, the **Workspace Explorer** is visible when you enter this task. The **Workspace Explorer** displays the contents of the workspace in a classification hierarchy that reflects the various relationships defined for the design objects. The content represents the current objects loaded from the database into the active workspace.



Icons display at the left of the window objects to indicate the type of the object. For example, a file folder icon represents the model, an equipment icon represents equipment, an I-beam icon represents a structural system, and so forth.

See Also

Interface Overview (on page 24)
Detail View (View Menu) (on page 29)
Management Console (View Menu) (on page 28)
Refresh (View Menu) (on page 32)

Refresh (View Menu)

Updates the *loaded* (expanded) content of the **Console** hierarchy, when others are concurrently adding folder, components, or drawings to the hierarchy. The entire hierarchy does not refresh unless you have all the nodes completely expanded. You can also press **F5** to update the content.

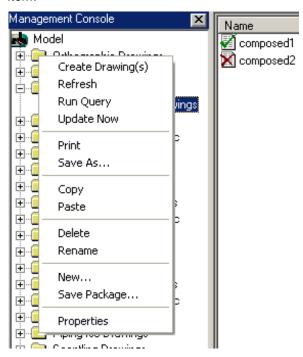
See Also

Shortcut Menus (on page 33)
Interface Overview (on page 24)
Detail View (View Menu) (on page 29)
Management Console (View Menu) (on page 28)

SECTION 5

Shortcut Menus

When you right-click nodes in the **Management Console** and nodes or documents in the **Detail View**, shortcut menus display. The items on the shortcut menu vary depending on the selected item.



Root Shortcut Menu (on page 34)
Folder Shortcut Menu (on page 34)
Component Shortcut Menu (on page 35)
Drawing Document Shortcut Menu (on page 36)

■ NOTES

- By default, the Management Console and Detail View display when you enter the task.
 You can turn the views on or off on the View menu.
- When you switch to a different task and then return to the Drawings and Reports task, the Management Console remembers the node you last selected on the console hierarchy.
- If you are registered with SmartPlant Foundation, the **Publish** and **Revise** commands are added as well. You can also set up batch printing and updating for documents using the **Batch** commands available on the shortcut menus.

See Also

Components Overview (on page 89)

Root Shortcut Menu

The Root shortcut menu displays the commands that you use to manage drawings at the Model level. It is available when you right-click the Model root node in the **Management Console**.

You must first create a new folder using the **New** command on the Folder shortcut menu. The commands of the Folder shortcut menu change after adding a folder.

Before Adding Folders

After Adding Folders

New (see "New Command" on page 41)

Properties (see "Properties Command" on page 48)

Create Drawing(s) Command (on page 39)

Refresh (see "Refresh (Shortcut Menu)" on page 79)

Run Query (see "Run Query (Shortcut Menu)" on page 70)

Update (on page 83) (Drawings by Rule in marine mode and material handling mode only)

Update Now (on page 80) (All other drawing types)

Batch (see "Batch Commands" on page 37)

- Print
- Update
- Update Document(s)
- Refresh

Print (see "Print Command" on page 45)

Save As Command (on page 70)

New (see "New Command" on page 41)

Properties (see "Properties Command" on page 48)

See Also

Shortcut Menus (on page 33)

Folder Shortcut Menu

The Folder shortcut menu displays the commands that you use to manage drawings at the folder level. It is available when you right-click a folder node in the **Management Console**.

After creating a new folder from the Root shortcut menu, you must first create a new component or another new folder using the **New** command on the Folder shortcut menu. The commands of the Folder shortcut menu change after adding an item to the folder.

Before Adding Folders or Components

After Adding Folders or Components

Batch (see "Batch Commands" on page 37)

Create Drawing(s) Command (on page 39)

Print

Refresh (see "Refresh (Shortcut Menu)" on

- Update
- Update Document(s)
- Refresh

Copy (see "Copy Command" on page 38)

Paste (see "Paste Command" on page 43)

Delete (see "Delete Command" on page 39)

Rename (see "Rename Command" on page 69)

New (see "New Command" on page 41)

Save Package (see "Save Package Command" on page 76)

Properties (see "Properties Command" on page 48)

page 79)

Run Query (see "Run Query (Shortcut Menu)" on page 70)

Update (on page 83) (Drawings by Rule in marine mode and material handling mode only)

Update Now (on page 80) (All other drawing types)

Batch (see "Batch Commands" on page 37)

- Print
- Update
- Update Document(s)
- Refresh

Print (see "Print Command" on page 45)

Save As Command (on page 70)

Copy (see "Copy Command" on page 38)

Paste (see "Paste Command" on page 43)

Delete (see "Delete Command" on page 39)

Rename (see "Rename Command" on page

New (see "New Command" on page 41)

Save Package (see "Save Package Command" on page 76)

Properties (see "Properties Command" on page 48)

See Also

Shortcut Menus (on page 33)

Component Shortcut Menu

The Component shortcut menu displays the commands that you use to manage drawings at the component level. It is available when you right-click a component node in the Management Console.

The commands of the Component shortcut menu change after creating drawings using the Create Drawing(s) command.

Before Creating Drawings

Setup (see "Setup Command" on page 78)

New (on page 37)

Create Drawing(s) Command (on page 39)

Update (on page 83) (Drawings by Rule in

After Creating Drawings

New (on page 37)

Create Drawing(s) Command (on page 39)

Refresh (see "Refresh (Shortcut Menu)" on page 79)

marine mode and material handling mode only)

Delete (see "Delete Command" on page 39)

Rename (see "Rename Command" on page 69)

Save As Command (on page 70)

Edit Template (see "Edit Template Command" on page 41)

Save Package (see "Save Package Command" on page 76)

Properties (see "Properties Command" on page 48)

Run Query (see "Run Query (Shortcut Menu)" on page 70)

Update (on page 83) (Drawings by Rule in marine mode and material handling mode only)

Update Now (on page 80) (All other drawing types)

Batch (see "Batch Commands" on page 37)

- Print
- Update
- Update Document(s)
- Refresh

Print (see "Print Command" on page 45)

Save As Command (on page 70)

Copy Command (on page 38)

Delete (see "Delete Command" on page 39)

Rename (see "Rename Command" on page 69)

Save Package (see "Save Package Command" on page 76)

Setup (see "Setup Command" on page 78)

Edit Template (see "Edit Template Command" on page 41)

Properties (see "Properties Command" on page 48)

See Also

Shortcut Menus (on page 33)

Drawing Document Shortcut Menu

The Drawing Document shortcut menu displays the commands that you use to manage drawings at the individual document level. It is available when you right-click a drawing in the **Detail View** of the **Management Console**.

After creating a new component from the Folder shortcut menu, you must first create drawings using the **Create Drawing(s)** command on the Model Root, Folder, or Component shortcut menus. The Drawing Document shortcut menu contains the following commands:

Open Command (on page 43)

Edit (see "Edit Command" on page 40)

Refresh (see "Refresh (Shortcut Menu)" on page 79)

View Log (see "View Log Command" on page 88)

Batch (see "Batch Commands" on page 37)

Update (on page 83) (Drawings by Rule in marine mode and material handling mode only)

Update Now (on page 80) (All other drawing types)

Rename (see "Rename Command" on page 69)

Delete (see "Delete Command" on page 39)

Print (see "Print Command" on page 45)

Save As Command (on page 70)

Properties (see "Properties Command" on page 48)

See Also

Shortcut Menus (on page 33)

New

Generates an empty drawing containing no views. This command is available on the right-click menu for a component in the **Management Console** hierarchy.

The empty drawing uses the layout styles assigned to the component. After you add drawing documents, you can open them and add unassigned views as needed.

TIP When the component uses document or sheet "No Assignment" rules in the **Tools** > **Define Layout Style** command, you can use the **New** command to create an empty drawing in a component without views, and manually add all views.

Add a drawing document

1. Right-click the component, and select New.

A drawing document using the document naming rule displays in the Detail View of the Management Console, or as a child of the component in the **Drawing Console**.

2. Repeat the **New** command to create the number of needed drawing documents.



■ NOTES

- Document Naming and Sheet Naming rules are customizable items.
- For more information about sheet properties and naming rules, see "Edit Sheet Command" in SmartSketch Drawing Editor Help.
- All unassigned views in SmartSketch Drawing Editor are available for placement on documents created with New command.

Batch Commands

Batch commands are only available on shortcut menus if batch processing and a batch processing server have been set up by an administrator. For more information, see Batch Processing.

Copy Command

Copies a drawing, component, or folder. After you copy an item, you can paste it at another location in the hierarchy.

The software enforces the following rules regarding pasting:

- If you copy a drawing, select a component to use the Paste command. You cannot paste a
 drawing to a folder or under the drawing book
 - **NOTE** You can copy only the composed drawings and Drawing by Rule drawings.
- If you copy a composed drawing and paste it on a Composed Drawing component, the Paste Special dialog box is displayed. This dialog box gives you the option to paste the drawing with or without the volume associated with the view. For more information, see Paste Special Dialog Box (on page 44).
 - NOTE The Paste Special dialog box is displayed for a composed drawing only if it is copied for drawings containing views with associated volumes.
- If you copy an application component, select a folder to use the **Paste** command. You cannot paste an application component under the drawing book.
- If you copy a folder, select a folder or a drawing book to use the Paste command.
- If you copy a volume or a MicroStation 3D DGN drawings component with a template or a drawing volume associated with it, the **Paste Special** dialog box is displayed if you paste the component in another volume or a MicroStation 3D DGN drawings component. This dialog box gives you the following options to paste: 1) only the template, (2) only the volume, or (3) both the template and the volume. The volume associated with the current volume drawings component being pasted, is copied and associated with the new volume drawings component. For more information, see *Paste Special Dialog Box* (on page 44).
 - NOTE A component cannot be pasted under another component.

Copy an item

- 1. Select an item in the Management Console hierarchy or Detail View.
- 2. Right-click the item, and then select **Copy** on the shortcut menu.
- NOTE You can paste the copied item at another location in the **Management Console** hierarchy. For more information, see *Paste an item* (on page 44).

See Also

Shortcut Menus (on page 33) Components Overview (on page 89)

Cut Command

Moves a drawing, component, or folder from its current location so that you can paste it at another location in the hierarchy.

The software enforces the following rules regarding pasting:

 If you cut a drawing, select a component to use the Paste command. You cannot paste a drawing to a folder or under the drawing book NOTE Only the composed drawings can be cut.

- If you cut an application component, select a folder to use the **Paste** command. You cannot paste an application component under the drawing book.
- If you cut a folder, select a folder or a drawing book to use the **Paste** command.

Cut an item

- 1. Select an item in the Management Console hierarchy or Detail View.
- 2. Right-click the item, and then select **Cut** on the shortcut menu.

■ NOTE You can paste the cut item at another location in the **Management Console** hierarchy. For more information, see *Paste an item* (on page 44).

See Also

Shortcut Menus (on page 33) Components Overview (on page 89)

Create Drawing(s) Command

Generates the drawings that have not previously been created. This command is available on the right-click menu for various items in the **Management Console** hierarchy.

If you select the top-level of the hierarchy, this command generates all drawings not already created for all components in the hierarchy. For example, if you have Volume Component drawings that have not been created and Isometric drawings that have not been created, both are created if you right-click the top-level hierarchy and select **Create Drawing(s)**.

You can also right-click individual components or folders for which drawings are not yet created and select **Create Drawing(s)** on the shortcut menu to generate the drawings.

After you create drawing documents, you update them to include model object content. You can then open or edit them as needed.

■ NOTE Marine mode or material handling mode only: If you did not previously select the model contents for a drawings-by-rule component, the Filter Properties for Asking Filter dialog box displays, and you must select a filter. The dialog is the same as the Filter Properties dialog box, except that only the appropriate tabs for the asking filter are displayed. For more information, see Filter Properties Dialog Box (on page 164).

See Also

Updating Documents (on page 78) Shortcut Menus (on page 33) New (on page 37) Run Query (Shortcut Menu) (on page 70)

Delete Command

Removes an item and its sub-items from the hierarchy and the database. You access this command on the right-click menu for any node or document in the hierarchy. **Undo** is not available for this action. Upon deletion, a confirmation message displays.

When you delete a drawing, its associated template and its component remain unchanged. Any associated physical volumes are deleted.

You can delete many items that are directly or indirectly related to this task:

- Components in the Management Console hierarchy
- Documents
- Drawing volumes
- Drawing views

In some cases, deleting an item causes other items to be deleted. For example, when you delete a drawing volume, associated views are deleted. When you delete a drawing view in a generated drawing, associated volumes and documents are deleted.

In addition, the item status and your permissions can affect whether or not you can delete the item. A drawing set to Approved cannot be deleted. However, you may be able to delete a drawing set to Working.

NOTE You can select multiple components or documents in the **Detail View** and use the **Delete** command from the right-click menu to remove those items.

Delete an item

- 1. Right-click a folder component, document, drawing volume, or drawing view.
- 2. On the shortcut menu, click Delete.
- **WARNING** You cannot undo a delete operation.

NOTE The **Delete** command propagates down the hierarchy. For example, if you delete a volume component, its child components (if any) and all the drawings contained in the components are deleted as well. However, when you delete a single volume drawing, the associated template, volume, and component remain unchanged. You can update the drawing component to re-create the drawing.

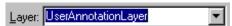
See Also

Shortcut Menus (on page 33) Components Overview (on page 89)

Edit Command

Activates the selected drawing for editing. This command is available on the right-click menu for a drawing in the **Detail View**. The drawing opens in **SmartSketch Drawing Editor** with additional commands or toolbars available for editing the drawing.

Objects like drawing views, key plan views, report views, and drawing property labels are placed on the **DwgTemplate** layer when you save the drawing document. You should not place manual markups on the **DwgTemplate** layer. If you use native **SmartSketch Drawing Editor** commands (such as **Place Line** or **Place Dimension**) to add manual markups to the template, put them on the **Default** layer or on a layer that you define (for example, a layer named **AnnotationLayer**). This preserves the changes when you update drawings. For more information on layers, see *Layers* (*SmartSketch Drawing Editor Tools Menu*) (on page 402).



Your access permissions, defined in the Project Management task, affect whether or not you can edit documents.

Edit Template Command

Allows you to open and edit a template for a drawing component in SmartSketch Drawing Editor.

New Command

Adds a new folder, a search folder, an imported folder, or a component to the **Management Console** hierarchy. Select the model root or a folder and right-click to select **New**. If you select the model root, **New** creates a folder, a search folder, or a folder containing a set of drawing components. If you select a folder, New creates a folder, a search folder, an imported folder, or a drawing component.

Search Folders (on page 232)

Imported Folders (on page 237)

Components Overview (on page 89)

If you have previously created a package, the package is available in the **Add Component** dialog box to add a component to the hierarchy. For more information, see *Save Package Command* (on page 76).

You can configure the shortcut menu for a folder to display more **New** options. For more information, see *Configure New Command On A Folder* in the *Smart 3D Drawings and Reports Reference Data Guide*.

Add Component Dialog Box (on page 43)

What do you want to do?

- Add a folder (on page 41)
- Add a package of components (on page 42)
- Add a component (on page 42)

Add a folder

 Right-click the root model in the Management Console hierarchy, and select New > Folder.

The folder is added to the hierarchy.

2. To rename the folder, right-click the folder, and select **Rename**, or select the folder, and press **F2** on the keyboard. Type a new name.

- You can place another folder under a folder in the hierarchy.
- You can place a folder, application component, or a package under a folder in the hierarchy.
 For example, you can add an Isogen Isometric Drawings by Query component to a folder.
- You cannot add folders or other components to a search folder component. For more information, see Search Folders (on page 232).

Add a package of components

- 1. Right-click the root model in the Management Console hierarchy, and select New.
- 2. On the shortcut menu, click **More** to open the **Add Component** dialog box, and select a package for a set of related components. For more information, see *Add Component Dialog Box* (on page 43).
 - A folder and components are added to the hierarchy.
- 3. To rename the folder or a component, right-click and select **Rename** on the shortcut menu, or press **F2** on the keyboard.
- **NOTE** Right-click a component to access the available commands for that component.

Add a component

- 1. In the **Management Console**, create a folder or select an existing folder.
- 2. Right-click the folder, then select New.
- On the shortcut menu, select a component such as Composed Drawings or Orthographic Drawings by Query, or click More to open the Add Component dialog box and select a component. For more information, see Add Component Dialog Box (on page 43).
 - The component is added to the folder.
- 4. To rename the component, right-click the component, and select **Rename**, or select the component, and press **F2** on the keyboard. Type a new name.

■ NOTES

- Right-click a component to access the available commands for that component.
- You cannot add folders or other components to a search folder component. For more information, see Search Folders (on page 232).

Add a Drawings by Rule component

This workflow is for marine mode and material handling mode only.

- 1. In the **Management Console**, create a folder or highlight an existing folder.
- 2. Right-click the folder, and select New.
- 3. On the shortcut menu, click More.
 - The Add Component dialog box displays.
- 4. Select a tab containing Drawings by Rule packages, such as **Ship**, **Manufacturing**, **Ship Structure**, **Piping**, or **Offshore**.
- 5. Select a Drawings by Rule package 🔄, and click **OK**.
 - The component is created in the folder.

- Right-click a component to access the available commands for that component.
- You cannot add folders or other components to a search folder component. For more information, see Search Folders (on page 232).

Add Component Dialog Box

Lists the available packages and folder types you can use to create folders and drawing components at the selected level in the **Management Console** hierarchy. You access this dialog box when you right-click a folder or the model root, and select **New > More**.

Packages are organized in tabs by drawing type. Select a tab, and then select a package to view its description.

■ NOTE Many packages are delivered with the software. Your catalog administrator can add new tabs and packages to the dialog box using the **Save Package Command**. For more information, see *Save Package Command* (on page 76).

See Also

Components Overview (on page 89)
Delivered Drawing Types (on page 21)
Icons for Components and Drawings (on page 25)

Open Command

Opens the selected document for viewing within this task. This command is available on the shortcut menu for all document types except 3D Model Data. You also can open the document by double-clicking it.

NOTE Opening a MicroStation 3D DGN document requires that the MicroStation J (V7) or MicroStation J (V8) application be loaded on the workstation.

Open a document

- 1. In the **Detail View**, double-click a document. You can also right-click the document, then select **Open**.
- 2. Close a document by clicking File > Exit.

■ NOTE To edit the document, right-click the document, the select **Edit**. If the document is a drawing, you can annotate it.

See Also

Open Command (on page 43) Shortcut Menus (on page 33) Components Overview (on page 89)

Paste Command

Inserts the contents that you last copied or cut into the hierarchy or, creates a copy of a drawing volume, depending on the component that you have selected. You must copy or cut an item using the **Copy** or **Cut** command before you can paste it.

The software enforces the following rules regarding pasting:

If you copy or cut a drawing, select a component to use the Paste command.

- You can copy only the Composed or the Drawing by Rule drawings.
- You can cut only the composed drawings.

- If you paste a composed drawing on a Composed Drawings component, the software displays the Paste Special dialog box. This dialog box gives you the option to paste the drawing with or without the volume associated with the view. For more information, see Paste Special Dialog Box (on page 44).
 - **NOTE** The **Paste Special** dialog is displayed only for the composed drawings if it is copied for the drawings containing views with associated volumes.
- If you copy a folder, select a folder or a drawing book to use the Paste command. You cannot paste a folder to an application component. Application components cannot have a folder below them in the hierarchy.
- If you copy an application component and the Paste command is selected on a folder, the copied application component is pasted under the folder.
- If the application component you copy is a volume drawing or a MicroStation 3D DGN component, and if the component has a template or a drawing volume associated with it, the Paste Special dialog box is displayed. The dialog box gives the option to paste the component with or without the template or the volume of that component.
- If you copy a volume drawings component with a template or a drawing volume associated with it and paste it in another volume drawings component, the software displays the **Paste Special** dialog box. This dialog box gives you the following options to paste: (1) only the template, (2) only the volume, or (3) both the template and the volume. For more information, see *Paste Special Dialog Box* (on page 44).
- If you copy a MicroStation 3D DGN component with a drawing volume associated with it, the software displays the **Paste Special** dialog box if you paste onto another MicroStation 3D DGN component.

Paste Special Dialog Box (on page 44)

Paste an item

Before pasting an item, you must copy or cut the item using the **Copy** or the **Cut** command. For more information, see *Copy an item* (on page 38) and *Cut an item* (on page 39).

- 1. Select a location in the **Management Console** hierarchy or **Detail View**.
- 2. Right-click the location, and click Paste.

The software pastes the item under the selected location.

See Also

Shortcut Menus (on page 33) Components Overview (on page 89)

Paste Special Dialog Box

Specifies the items to paste if you copied a volume drawing or MicroStation 3D DGN component that has a template or drawing volume defined. The options provided to you depend on the component selected.

If you are pasting a hierarchy containing one or more volume drawings or MicroStation 3D DGN components into a folder, you can select one of the following options:

Copy Nodes(s), Template(s), and Volume(s)

Inserts the new components, including their respective template and drawing volumes, under the selected folder.

Copy Node(s) and Template(s)

Inserts the new components, including their respective templates, under the selected folder.

Copy Node(s) Only

Inserts the new components under the selected folder.

If you are pasting a volume drawings component onto another volume drawings component, or a MicroStation 3D DGN component onto another MicroStation 3D DGN component, you can select one of the following options:

Copy Template(s), and Volume(s)

Copies the template settings and drawing volumes to the selected component.

Copy Template(s) only

Copies only the template settings to the selected component.

Copy Volumes(s) only

Copies only the drawing volumes to the selected component.

If you are pasting a composed drawing on a Composed Drawing snap-in, the following options are available:

Drawing(s) only

Copies only the drawing and its views

Drawing(s) and Associated Volume(s)

Copies the volume associated with the view along with the drawing.

See Also

Paste Command (on page 43)

Print Command

Sends a print request for the selected documents to the default printer. This command is not available until you have created and updated documents.

What do you want to do?

- Print a document (on page 45)
- Print a document as a PDF file (on page 46)

Print a document

- 1. To specify a printer, select **File** > **Select Printer**. For more information on selecting a printer, see *Select Printer (File Menu)* (on page 46).
- Select a folder, application component, or the root node in the Management Console to
 print all of the drawing and report documents beneath the selected level. You can also
 select a single document or multi- select documents in the Detail View. You can select
 multiple documents to print by pressing Ctrl or Shift and then clicking each document in the
 Detail View.
- Right-click and select **Print** on the shortcut menu.

You can also use the **Batch** > **Print** command on the item's shortcut menu to print the item using a scheduled batch job. For more information, see Batch Print.

Print a document as a PDF file

- 1. To specify a printer, select **File** > **Select Printer**. For more information on selecting a printer, see *Select Printer (File Menu)* (on page 46).
- 2. Select SmartPlant PDF Converter 4xx as the printer.
- 3. Select a folder, application component, or the root node in the Management Console to print all of the drawing and report documents beneath the selected level. You can also select a single document or multi-select documents in the Detail View. You can select multiple documents to print by pressing Ctrl or Shift and then clicking each document in the Detail View.
- 4. Right-click and select **Print** on the shortcut menu.

You can also use the **Batch** > **Print** command on the item's shortcut menu to print the item using a scheduled batch job. For more information, see Batch Print.

Select Printer (File Menu)

Specifies a printer for documents. The command lists all printers available to your computer.

■ NOTES

- To print a drawing as a PDF, select SmartPlant PDF Converter 4xx as the printer.
- SmartPlant PDF Converter 4xx requires special settings for 64-bit versions of Windows.
 For more information, see Configure SmartPlant PDF Converter for Windows (on page 46).

See Also

Select Printer Dialog Box (on page 47)

Configure SmartPlant PDF Converter for Windows

Smart 3D installs **SmartPlant PDF Converter 4***xx*, a printer driver used to convert drawings and reports to PDF files. Special settings are needed for this driver.

Windows 7

1. Open Start > Control Panel

The Control Panel displays.

- 2. In the Hardware and Sound section, click View devices and Printers.
- 3. Right-click SmartPlant PDF Converter 4xx and select Printer properties.

The **SmartPlant PDF Converter 4xx Properties** dialog box displays.

- 4. On the Advanced tab:
 - Select Spool print documents so program finishes printing faster.
 - Select Start printing immediately.
 - Clear Enable advanced printing features.
- Click OK.

Select Printer Dialog Box

Specifies a printer for documents.

Name

Specifies a printer name.

Status

Displays the current status of the specified printer.

Type

Displays the type of printer.

Where

Displays the port or location the printer uses.

See Also

Components Overview (on page 89)
Print a Document (on page 45)

Setting Properties

The software updates properties from parent nodes to child nodes and drawings in the **Management Console** hierarchy.

For example, you can display the **Properties** dialog box for a folder named **Isometric Drawings**. If you set the **Division Location** property to **Huntsville**, **Alabama**, the software pushes this value to the items contained within the **Isometric Drawings** folder.

You can specify inheritance for each item on its **Properties** dialog box. If you set the override flag for a property, the property is not inherited from the parent. You can provide a new, overriding value for the property. This new value then propagates to other items deeper in the hierarchy.

Properties and Publishing

Several document properties impact publishing the document. Before you can publish documents in the software, you must configure your computer. The configuration includes installing the SmartPlant Client and SmartPlant Schema Component and registering through the SmartPlant Registration Wizard. For more information about the SmartPlant Registration Wizard, see the *Intergraph Smart*TM 3D *Installation Guide*, available from **Help** > **Printable Guides**.

Even if you have registered your model using the SmartPlant Registration Wizard, you must set certain properties to enable the publishing capability. Properties that control publishing are found on the **WBS Tab**. For more information, see *Set properties for publishing documents* (on page 428).

- The software considers blanks or cleared values as override flags.
- If the drawing document you are looking at in the **Detail View** has a yellow icon (for example:), the drawing document is a version 6.1 legacy Snapshot drawing. You should use the **Tools** > **Convert Legacy Snapshots** command to convert this document to a

Composed Drawing for use in the current version of the software. If you do not convert the legacy snapshot drawing, you cannot perform edit operations on the drawing, including update, revise, and publish.

See Also

Edit document properties (on page 48)
Properties Command (on page 48)
Convert Legacy Snapshots (on page 240)

Properties Command

Views and edits properties for the selected document. The properties of child items are inherited from the parent item unless you set the **Override** column for the properties.

Properties Dialog Box (on page 49)

What do you want to do?

- Edit document properties (on page 48)
- Set surface styles and aspects for 3D model data documents (on page 48)
- Set properties for publishing documents (on page 428)

Edit document properties

1. In the **Management Console** or **Detail** view, right-click an item in the hierarchy, and select **Properties** on the shortcut menu.

The **Properties** dialog box displays.

2. Change the properties as needed. For example, you can set the coordinate system for the drawing on the **Style** tab or view the current approval information on the **Signature** tab.

■ NOTES

- If you do not want an item to acquire a property from its parent, select the **Override** column on the **Properties** dialog box and type a new value. This value propagates to other items deeper in the hierarchy.
- The software treats blank or cleared property values as overrides.

Set surface styles and aspects for 3D model data documents

Surface Style Rules and **Aspects** must be set on each 3D Model Data document before updating it using the **Update**, **Update Now**, or **Batch** > **Update** command.

- 1. Select one or more 3D Model Data documents.
- 2. Right-click the selected documents, and select **Properties** on the shortcut menu.

The **Properties** dialog box displays.

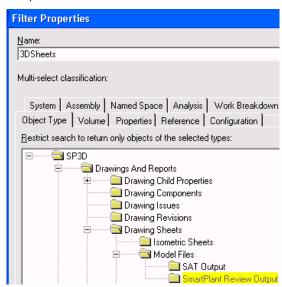
3. Go to the **Surface Styles and Aspects** tab to add surface style rules to the selected rules list. You can also import the surface styles used in the session file. For more information, see *Surface Styles and Aspects Tab (Properties Dialog Box)* (on page 56).

- 4. Click Select Aspects to specify the aspects used within the 3D Model Data documents. If you select no aspects for your documents, the Simple Physical aspect is automatically applied by default. For more information, see Select Aspects Dialog Box (on page 58).
- 5. Click **OK** on the **Properties** dialog box to save the changes to the document(s).

Now you can update the drawing documents using the **Update Now** or **Batch** > **Update** command to incorporate the surface styles and aspects with the 3D Model Data.

Use a Search Folder to Collect 3D Model Data Documents for Property Update

You can create a Search Folder that filters for the 3D Model Data documents so you can edit their surface style rule and aspect settings collectively. When you setup the **Search Folder**, use a filter that looks for the **SmartPlant Review Output** objects. You could additionally specify properties on the output objects to further narrow the search criteria, such as **Data Created**, **Date Last Modified**, or **Signature**. For more information, see *Search Folder Filters* (on page 234).



Properties Dialog Box

Sets options for items in the **Management Console** hierarchy. All items in the **Management Console** have a **Properties** command on their right-click shortcut menus. Using the **Properties** dialog box, you can control how you want properties to propagate through the hierarchy.

You can specify labels for some of the properties on the tabs. Click the browse button at the right of the table cell to display the **Choose Label** dialog box.

General Tab (Properties Dialog Box) (on page 50)

Title Area Tab (Properties Dialog Box) (on page 50)

Signature Area Tab (Properties Dialog Box) (on page 53)

Style Tab (Properties Dialog Box) (on page 55)

Surface Styles and Aspects Tab (Properties Dialog Box) (on page 56)

Custom Tab (Properties Dialog Box) (on page 62)

Notes Tab (Properties Dialog Box) (on page 63)

Issue Tab (Properties Dialog Box) (on page 64)

Revision Tab (Properties Dialog Box) (on page 65)

WBS Tab (Properties Dialog Box) (on page 67) Configuration Tab (Properties Dialog Box) (on page 68)

■ NOTES

- The WBS tab is available only if you have registered your model using the SmartPlant Registration Wizard.
- If your model has not been registered using the SmartPlant Registration Wizard, the **Issue** tab is always available. If your model has been registered, the **Issue** tab is available only if you have issued documents to a contract and the information is read-only. For more information, see *Issue request documents* (on page 434).

See Also

Setting Properties (on page 47)

General Tab (Properties Dialog Box)

Shows general properties for a drawing item.

Name

Displays the name of the property.

Value

Sets the current value of the property.

Behavior

Specifies whether to inherit or override a property in the hierarchy of items in the **Console**. If the property is **Inherited**, the setting comes from items higher in the hierarchy. If the property is not read-only, you can provide a new, overriding value for the property by changing the behavior setting to **Override**. This new value then propagates to other items deeper in the hierarchy. If you set the behavior to **Force Override**, the property setting is forced to items deeper in the hierarchy, even if they are set to **Inherit**.

Properties

Published

Indicates whether the drawing has been published.

Size

Displays the size of the file in KB (kilobytes).

See Also

Properties Dialog Box (on page 49)

Title Area Tab (Properties Dialog Box)

Sets options for the title area of drawings.

If you access this tab after selecting multiple drawings, these fields will appear empty, regardless of what information was defined for any of the drawings individually. Any information you add to this tab will overwrite the previously defined information in corresponding rows in the selected drawings.

Name

Displays the name of the property.

Value

Sets the current value of the property.

Behavior

Specifies whether to inherit or override a property in the hierarchy of items in the **Console**. If the property is **Inherited**, the setting comes from items higher in the hierarchy. If the property is not read-only, you can provide a new, overriding value for the property by changing the behavior setting to **Override**. This new value then propagates to other items deeper in the hierarchy. If you set the behavior to **Force Override**, the property setting is forced to items deeper in the hierarchy, even if they are set to **Inherit**.

Properties

Border

Specifies the border attribute that stores the name of border for the title block. This attribute also stores the dimensions of the border.

Charge Number

Defines the charge number for the drawing.

Charge Title

Describes the **Charge Number** box. The charge title text is placed to the left of the charge number.

Company Name

Specifies the name of the company for which the project is designed.

Desc1

Describes the drawing. This description is the first of four lines of text used to describe the drawing.

Desc2

Describes the drawing. This description is the second of four lines of text used to describe the drawing.

Desc3

Describes the drawing. This description is the third of four lines of text used to describe the drawing.

Desc4

Describes the drawing. This description is the fourth of four lines of text used to describe the drawing.

Division Location

Specifies the location of the division for which the project is designed.

Division Name

Specifies the name of the division for which this project is designed.

Drawing Naming Rule

Specifies a default or custom naming rule to the drawing. The default name rules provided include:

Default Drawing Name Rule - This is the default rule for composed drawing types. Creates a name based on the parent component name, Global Workshare location ID, and an index number. The naming rule inserts a "-" between each name. If there are no parent objects, then only the child object name is used. For example, the first drawing created under the ComposedDrawings001 component at workshare site 1 is called *ComposedDrawings001-1-0001*.

Default By Query Name Rule - This is the default rule for all Drawings by Query drawing types. Creates a name based on the parent object name (each system and sub-system above the child object will be included in the name) and child object name. The naming rule inserts a "-" between each name. If there are no parent objects, then only the child object name is used. For example, the first composed drawing created at workshare site 1 under the ComposedDrawings001 component would be named *ComposedDrawings001-0001*.

Default Report Name Rule - This is the default rule for all Spreadsheet Reports. Creates a name based on the object name, Global Workshare location ID, and an index number. For example, the first drawing created at workshare site 1 is called *ComposedDrawings001-1-0001*.

Volume Name Rule - This is the default rule for volume drawing types. Creates a name based on the volume name, Global Workshare location ID, and an index number. For example, the first drawing created using the volume Volume001 in workshare site 1 is called *Volume001-1-0001*. If a volume is not specified, the drawing name will be named *Unspecified*.

NOTE Customized naming rules appear in the list if you bulkload against the **CDrawingSheet** class, which is the class for the drawing object. The rules are defined on the **NamingRules** sheet in the *GenericNamingRules.xls* workbook. For more information, see the *Reference Data Guide*.

Drawing Number

Displays the unique identifier for the drawing.

Drawing Size

Defines a standard note value for the border size.

Drawing Type

Defines the three-letter code to identify the type of drawing. For example, the type can be DGN.

Job Number

Defines the unique identifier assigned to a capital project or job.

Note Line

Specifies text for a miscellaneous note line.

Plant Name

Specifies the name of the plant or project for which the drawing is designed.

Project Version

Defines the number and letter sequence that identifies a particular generation of a document that was created since the last approved revision.

Revision Numbers

Defines the number of the current revision for this drawing.

Sheet

Defines the number of the page and the total number of pages that are associated with this one. For example, the value might be **3 of 5**.

Site Location

Specifies the site location for which the drawing is designed.

Site Name

Specifies the name of the site where the plant is being constructed.

Title1

Specifies text for the first miscellaneous title. This title is usually a description of the area shown on the drawing.

Title2

Specifies text for the second miscellaneous title. This title is usually a description of the type of drawing.

Title3

Specifies text for the third miscellaneous title.

See Also

Properties Dialog Box (on page 49)

Signature Area Tab (Properties Dialog Box)

Sets options for the signature area of drawings.

If you access this tab after selecting multiple drawings, these fields will appear empty, regardless of the information defined for any of the drawings individually. Any information you add to this tab overrides the previously defined information in corresponding rows in the selected drawings.

Name

Displays the name of the property.

Value

Sets the current value of the property.

Behavior

Specifies whether to inherit or override a property in the hierarchy of items in the **Console**. If the property is **Inherited**, the setting comes from items higher in the hierarchy. If the property is not read-only, you can provide a new, overriding value for the property by changing the behavior setting to **Override**. This new value then propagates to other items deeper in the hierarchy. If you set the behavior to **Force Override**, the property setting is forced to items deeper in the hierarchy, even if they are set to **Inherit**.

Properties

Approval Date

Specifies the date the drawing was approved.

Approved By

Specifies the name of the person responsible for approving the drawing.

Checked By

Specifies the name of the person responsible for checking the drawing.

Checked Date

Specifies the date the drawing was checked.

Designed By

Specifies the name of the person who specified or designed the information on the drawing.

Designed Date

Specifies the date the drawing was designed.

Drawing Status

Defines the status code for the drawing.

Drawn By

Specifies the name of the person who drew the drawing, or created it.

Drawn Date

Specifies the date the drawing was drawn or created.

Extra Sign By1

Specifies the name of an extra person who is signing the drawing.

Extra Sign By2

Specifies the name of an extra person who is signing the drawing.

Extra Sign Date1

Specifies the date the drawing was signed by the person whose name displays in this box.

Extra Sign Date2

Specifies the date the drawing was signed by the person whose name displays in this box.

Extra Sign Title1

Defines the title of the person whose name displays in this box.

Extra Sign Title2

Defines the title of the person whose name displays in this box.

Mfg Rep Date

Specifies the date that the manufacturing representative initials the drawing.

Mfg Rep Name

Specifies the name of the manufacturing representative who signed the drawing.

Plant Number

Defines the plant number.

Proj Engineer Date

Specifies the date the project engineer initials the drawing.

Proj Engineer Name

Specifies the name of the person who is the project engineer for the project using the drawing.

Spec By

Specifies the name of the person who specified or designed the information on this drawing.

Spec Date

Specifies the date this drawing was specified or designed.

See Also

Properties Dialog Box (on page 49)

Style Tab (Properties Dialog Box)

Sets options for the style of drawings and reports.

If you access this tab after selecting multiple drawings, these fields will appear empty, regardless of the information defined for any of the drawings individually. Any information you add to this tab overrides the previously defined information in corresponding rows in the selected drawings.

Name

Displays the name of the property.

Value

Sets the current value of the property.

Behavior

Specifies whether to inherit or override a property in the hierarchy of items in the **Console**. If the property is **Inherited**, the setting comes from items higher in the hierarchy. If the property is not read-only, you can provide a new, overriding value for the property by changing the behavior setting to **Override**. This new value then propagates to other items deeper in the hierarchy. If you set the behavior to **Force Override**, the property setting is forced to items deeper in the hierarchy, even if they are set to **Inherit**.

Properties

Coordinate System

Specifies the global or an active coordinate system. Choose a coordinate system from the list, or click **More...** to choose another coordinate system with the **Select System** dialog box.

If you want to output large coordinates on your drawings, define a coordinate system using large negative coordinates. For example, if you want coordinates of **400,000 ft** output on drawings, define a coordinate system origin of **-400,000 ft** and place your model elements close to **global 0**. Select the new coordinate system in the **Coordinate System** field on the **Style** tab. For more information on defining coordinate systems, see the *Grids User's Guide* available from **Help > Printable Guides**.

■ NOTES

- The Coordinate System property is not used by Composed Drawings. The coordinate system settings are driven by the properties for the drawing views in a composed drawing. For more information on the coordinate system properties for a drawing view, see Drawing View Properties Dialog Box (Place View Command) in the SmartSketch Drawing Editor Help.
- If you are accessing the **Properties Style** tab for a 3D Model Data component or document with the intention of saving it as a SmartPlant Review file, make sure this property is set appropriately so that the **Plant Monument Coordinate Offset** is passed correctly to SPR when creating the VUE file. This is because SPR shows the objects from the VUE file using global coordinates. The offset value allows you to see the original coordinates relative to the new SPR coordinate system. For more information on 3D Model Data components, see 3D Model Data (on page 188). For information on saving to SPR, see Save as SmartPlant Review File (on page 417).

Volume Naming Rule

Specifies the naming rule applied to the content of the drawing.

Change Management

Enables and disables **Change Management** for Isogen isometric drawings. You can override the **Drawing.Content.ChangeManagementEnabled** option in the isometric drawing style with this property. You can set the property to **Enabled**, **Disabled**, or **Undefined**. This property is only available when you are viewing properties for an Isometric piping isometric drawing. For more information on Change Management, see Change Management in Piping Isometric Drawings.

Baseline Date

Identifies a date in time when a *snapshot* of the drawing document was taken. It is a way of identifying when objects have changed.

WBS Project

Specifies the Work Breakdown Structure (WBS) project style to be used with the drawing. This property serves as the answer to an asking filter when specified in a **View Style**. For more information, see *Create a Drawing Using WBS Objects* (on page 208).

See Also

Properties Dialog Box (on page 49)

Surface Styles and Aspects Tab (Properties Dialog Box)

Sets options for the surface styles and aspects used in 3D Model Data. This tab is only available when you are looking at the properties for a 3D Model Data document.

You can perform the following operations on this tab:

- Select an existing surface style rule from the library and add it to the workspace.
- Modify an existing surface style rule in the library and add it to the workspace.
- Create a new surface style rule and add it to the library and the workspace.
- Delete a surface style rule from the library or the workspace.
- Rearrange the style rules in the workspace box of the Surface Style Rules dialog box by using the Move Up and Move Down commands.

- Import surface styles from the session file.
- Set the aspects for the 3D Model Data.

Surface style rules

Style rule library

Lists all the current surface style rules in the Site database.

Selected rules

Lists all the names for the surface style rules currently assigned to the workspace.

Add

Adds the selected surface style rule to the workspace.

Remove

Removes a selected surface style rule from the workspace. To remove a surface style from the workspace, select the style in the **Workspace** list and click **Remove**.

Move Up

Moves the selected style rule up one step in the **Workspace** list.

Move Down

Moves the selected style rule down one step in the Workspace list.

New

Activates the **Surface Style Rule Properties** dialog box on which you can create a new surface style rule and add it to the database. This button is available only if you have write permission to the surface style rules.

Modify

Activates the **Surface Style Rule Properties** dialog box to modify an existing surface style rule and add it to the database.

■ NOTE For more information on creating new or modifying existing surface style rules, see *Surface Style Rule Properties Dialog Box* (on page 60).

Copy

Creates a copy of the selected rule on the Clipboard. You use **Copy** to create a copy of a surface style rule in the Model database so you can modify the rule rather than create a new one.

NOTE If you try to copy a style rule associated with a deleted filter, the style cannot be copied. A message box displays.

Paste

Pastes the copied rule from the Clipboard so it can be modified.

Delete

Removes the selected Surface Style Rule from the database.

Apply

Applies changes in surface style rules to the workspace.

NOTE Double-clicking a surface style rule also activates the Surface Style Rule

Properties dialog box on which you can create or modify a surface style rule if you have permission.

Import from Session

Imports session surface style rules for the selected 3D Model Data component.

Aspects

Select Aspects

Opens the **Select Aspects** dialog box so you can specify the aspects to use for the 3D Model Data component. For more information, see *Select Aspects Dialog Box* (on page 58).

NOTE When publishing 3D Model Data documents, the **Simple Physical** aspect is used by default if no other aspects are specified.

See Also

Properties Dialog Box (on page 49) Set surface styles and aspects for 3D model data documents (on page 48)

Select Aspects Dialog Box

Specifies the aspect associated with the 3D Model Data document(s). An aspect is a geometric area or space related to an object. The aspect represents information about the object, such as its physical shape or the space required around the object. Aspects are associated parameters for an object, representing additional information needed for placement. Aspects can represent clearances for safety or maintenance, additional space required during operation, or simple and detailed representations of the object. You define aspects when you model a part class for the reference data.

This dialog box displays when you click the **Select Aspects** button on the **Surface Styles and Aspects** tab of the **Properties** dialog box for a 3D Model Data document(s).

Selected aspects

Shows a checkbox list of all aspects defined by the model reference data. You can check multiple aspects. By default, the **Simple Physical** aspect is selected.

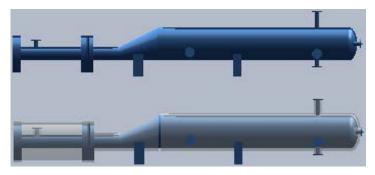
Simple physical - Includes primitive shapes. This aspect creates a less cluttered view
of the object, showing only the body of equipment or a simplified cross-section for
structure, as shown below.



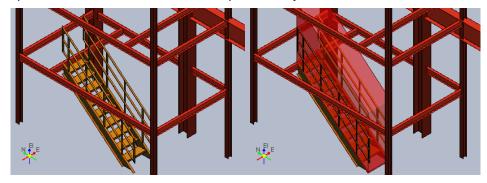
Detailed physical - Provides a more detailed view of an object. This aspect shows all the graphical details associated with the equipment or structure. For example, certain types of equipment may include legs and lugs. For marine structure, this aspect uses all geometry in the cross-section, as shown below.



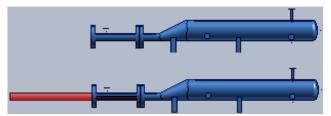
 Insulation - Shows an area around a piece of equipment indicating the presence of insulation. This aspect is also used to display structural fireproofing insulation. For example, a 4-inch pipe with insulation might look like an 8-inch pipe when this aspect is used.



Operation - Includes the area or space around the object required for operation of the object. This space shows in the model but not in drawings. For example, this aspect leaves enough space around a motor for a person to operate the motor or the overhead space needed for someone to walk up a stairway.



Maintenance - Includes the area or space around the object required to perform
maintenance on the object. This space may appear in the model but not in drawings.
For example, this aspect leaves enough space around a motor to perform maintenance
on a motor, including space to remove the motor.



Reference Geometry - Allows you to construct or add graphical objects that do not participate in interference checking. For example, a reference geometry object could be a spherical control or the obstruction volume for a door. For marine structure, this aspect allows you to control the display of landing curves for design seams, profile systems, reference curves, and knuckle curves.

Centerline - Displays objects as a single line representation. For example, this aspect is
useful for when you want to display handrails or structural members as a single-line on
drawings. For structure, the centerline is determined from the cardinal point used to
place the member. That cardinal point is not always the center of the object.



- Molded Forms Displays plate, profile, and beam systems for marine structure.
- Equipment Hole Displays holes for marine structure created in the Hole Management task.
- * Matches all cross-sections.

For more information on defining aspects for your model reference data, see the *Common User's Guide*.

See Also

Surface Styles and Aspects Tab (Properties Dialog Box) (on page 56) Set surface styles and aspects for 3D model data documents (on page 48)

Surface Style Rule Properties Dialog Box

Selects a filter and a surface style to be used for the objects identified by the filter. This dialog box displays when you click **New** or **Modify** or double-click a surface style rule on the **Surface Style Rules** dialog box. You can also use this dialog box to rename a rule after you use the **Copy** and **Paste** commands on the **Surface Style Rules** dialog box. **Paste** creates a rule named **Copy of original surface style rule name**.

Surface Style Rule Properties Tab (Surface Style Rule Properties Dialog Box) (on page 60) Configuration Tab (Surface Style Rule Properties Dialog Box) (on page 62)

Surface Style Rule Properties Tab (Surface Style Rule Properties Dialog Box)

Creates or modifies a surface style rule. Surface style rules are based on filters. When you create new rules or modify rules, you specify a filter on which to base the rule. For more information on filters, see the *Common User's Guide*.

Rule name

Specifies the name of the surface style rule.

Filter

Identifies the filter used within the style rule. The filters available are the ones defined for the current database. The list in the dropdown includes the last ten filters selected. Selecting **Create New Filter** in the dropdown list displays the **New Filter Properties** dialog box, which allows you to define a new filter for the style rule. Selecting **More** in the list displays the **Select Filter** dialog box. The **Properties** button for this field displays the **Property** dialog

box for the selected filter. For more information on defining a new filter or reviewing properties, see the *Common User's Guide*.

STIP We recommend that you use simple, asking, and compound filters with style rules. Using SQL filters can result in significant performance degradation and should be avoided whenever possible. Unlike the other types of filters, the SQL server is performed directly on the database. For each object passed to the SQL filter, the software checks to see if any of the objects were returned by the query. However, modification of the object may change whether or not the object passes the SQL filter. For example, a pipeline might pass the SQL filter before it is assigned to a different system. After the system assignment changes, a different style rule is applied. Therefore, some SQL filters may result in decreased efficiency in assessing the project data model.

Style applied

Specifies the surface style to be used for the objects identified by the selected filter. The list in the dropdown includes all surface styles available for the current database. The **Properties** button displays the **Surface Style Rule Properties** dialog box so you can edit the style as needed.

Select all aspects to which the style will be applied

Shows a checkbox list of all aspects defined by the model reference data. You can check multiple aspects. By default, all aspects are selected.

- An aspect is a geometric area or space related to an object. The aspect represents information about the object, such as its physical shape or the space required around the object. Aspects are associated parameters for an object, representing additional information needed for placement. Aspects can represent clearances for safety or maintenance, additional space required during operation, or simple and detailed representations of the object. You define aspects when you model a part class for the reference data.
- The Simple Physical aspect includes primitive shapes. The space can be a field junction box displayed in both the model and in drawings. When you publish 3D Model Data documents, this is the default aspect used if no other aspects are selected for the document properties.
- The Detailed Physical aspect provides a more detailed view of equipment in the model. For example, certain types of equipment may include legs and lugs. As opposed to the Simple Physical aspect, which only shows the body of the equipment, the Detailed Physical aspect shows all the graphical details associated with the equipment.
- The Insulation aspect shows an area around a piece of equipment indicating insulation is present. For example, a 4-inch pipe with insulation might look like an 8-inch pipe when the Insulation aspect is selected.
- The Operation aspect includes the area or space around the object required for operation of the object. This space shows in the model but not in drawings. The Operation aspect leaves enough space around a motor for a person to operate the motor.
- The Maintenance aspect includes the area or space around the object required to perform maintenance on the object. This space may appear in the model but not in drawings. The Maintenance aspect leaves enough space around a motor to perform maintenance on the motor, including space to remove the motor, if necessary.
- The Reference Geometry aspect allows you to construct or add graphical objects that do not participate in interference checking. For example, a reference geometry object could be

the obstruction volume for a door on a field junction box. Another example is a spherical control point.

See Also

Surface Style Rule Properties Dialog Box (on page 60)

Configuration Tab (Surface Style Rule Properties Dialog Box)

Displays the creation, modification, and status information of an object.

Model

Displays the name of the model. You cannot change this value.

Permission Group

Specifies the permission group to which the object belongs. You can select another permission group, if needed. Permission groups are created in the Project Management task.

Transfer

Re-assigns ownership of the selected model objects from their current permission group to another satellite or host permission group. This button is only available if the active model/project is replicated in a workshare configuration. The button is not available if all of the objects in the select set already belong to another location and are non-transferable. For more information, see *Transfer Ownership Dialog Box* in the *Structural Detailing User's Guide*.

NOTE The Transfer option does not apply to the Surface Style Rules.

Status

Specifies the current status of the selected object or filter. Depending on your access level, you may not be able to change the status of the object.

Created

Displays the date and time that the object was created.

Created by

Displays the name of the person who created the object.

Modified

Displays the date and time when the object was modified.

Modified by

Displays the name of the person who modified the object.

Custom Tab (Properties Dialog Box)

Sets options for drawing properties.

If you access this tab after selecting multiple drawings, these fields appear empty, regardless of the information defined for any of the drawings individually. Any information you add to this tab overrides the previously defined information in corresponding rows in all selected drawings.

Name

Displays the name of the property.

Value

Sets the current value of the property.

Behavior

Specifies whether to inherit or override a property in the hierarchy of items in the **Console**. If the property is **Inherited**, the setting comes from items higher in the hierarchy. If the property is not read-only, you can provide a new, overriding value for the property by changing the behavior setting to **Override**. This new value then propagates to other items deeper in the hierarchy. If you set the behavior to **Force Override**, the property setting is forced to items deeper in the hierarchy, even if they are set to **Inherit**.

See Also

Properties Dialog Box (on page 49)

Notes Tab (Properties Dialog Box)

Sets notes for the item.

If you access this tab after selecting multiple drawings, these fields appear empty, regardless of the information defined for any of the drawings individually. Any information you add to this tab overrides the previously defined information in corresponding rows in the selected drawings.

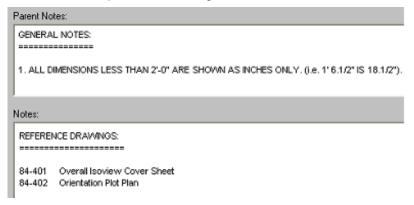
Parent Notes

Concatenates the notes from any parents of the currently selected item and displays the notes. This box is read-only.

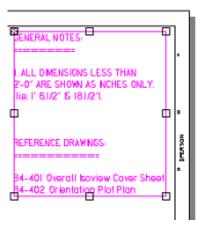
Notes

Specifies notes for the currently selected item.

The following graphics demonstrate how parent notes and notes work. The first graphic shows how notes can be added at different levels of the hierarchy. The **Notes** tab shows Parent Notes from a higher-level folder or component. The Notes section shows additional information for a particular drawing document.



When the document is updated and displayed, the Note Region of the template contains the specified information.



See Also

Properties Dialog Box (on page 49)

Issue Tab (Properties Dialog Box)

Sets options for internal issues.

Issue Number

Shows the drawing issue number. Type the required issue number for the drawing. When you have selected multiple drawings or a node in the **Management Console**, this field is not available to edit, as an issue number is created for each individual drawing, depending on its current revision history.

Description

Describes briefly the scope of the issue.

Issue Date

Shows the date issued.

Issue Reason

Shows the reason the document was issued. Select a value from **Bid**, **Fabrication**, **Construction**, **Reference**, and your customized values, if any are defined.

■ NOTE The values for Issue Reason can be customized by editing and bulkloading the DrawingIssueReason_Codelist.xIs workbook delivered in the [Product Folder]\CatalogData\Bulkload\AdditionalDataFiles folder. For more information, see Bulkload Files in the Drawings and Reports Reference Data Guide.

Job Spec

Identifies the job specification for the issued document.

Revision Number

Defines the revision number for this issue of the drawing.

Unregistered

If you access **Properties** on a single document when your model has not been registered using the SmartPlant Registration Wizard, the **Issue** tab displays previous entries. A new row is available to make a new entry. You can edit each field using alphanumeric and special characters. You cannot delete a row after it has been added.

If you access **Properties** on a folder when your model has not been registered using the SmartPlant Registration Wizard, the **Issue** tab has a single blank row for a new entry. With the exception of the **Issue Number** field, you can edit all the fields. Their values are propagated to the documents within the folder.

Registered

If your model is registered using the SmartPlant Registration Wizard and you have issued requests for the document, the **Issue** tab is read-only. The Issue information is retrieved for informational purposes only.

■ NOTES

- You can create only one issue per instance of the **Properties** dialog box. To create another issue, close the dialog box and open it again.
- For information on issuing requests for contracts when working in an integrated environment, see *Issue request documents* (on page 434).

See Also

Properties Dialog Box (on page 49)

Revision Tab (Properties Dialog Box)

Displays and sets properties for revisions. The **Revision** tab is always read-write (subject to user permissions).

Action

Lists the available actions when accessing Properties on a single component, multiple components, or multiple documents.

Append Record - Creates a new revision record for each document under the selected components or in the document set. You can type a value for the next revision mark or let the software automatically increment it for you.

Edit Last Record - Edits the last revision for each document under the selected components or in the document set. Only the edited revision fields overwrite the corresponding fields on the last revision record. To clear a populated revision field, type a single space character, and no other characters, in the edited field.

■ NOTE The Append Record and Edit Last Record options are not available for a model registered with SmartPlant Foundation or when revising a single document.

Revision Mark

Specifies the current revision. For single documents, double-click the **New Record** cell to automatically increment to the next revision mark number. To manually type a value for the next revision mark, click the **New Record** cell and type the value. This only applies when the model has not been registered with SmartPlant Foundation. If this cell is not edited, then the revision mark number automatically increments to the next available number in each writeable document associated to the selected set.

Revision Minor Number

Specifies the minor revision number for the revision.

Description

Describes the scope of the revision.

Revised By

Identifies the person who made the revision.

Revision Date

Specifies the date of the revision.

Check

Identifies the person who checked the revision.

Check Date

Specifies the date the revision was checked.

Approved By

Identifies the person who approved the revision.

Approval Date

Specifies the date the revision was approved.

The appearance and behavior of the contents of this tab differ depending on whether properties are accessed on a single document or accessed on a single component, multiple components, or multiple documents. The contents of this tab also depend on whether the model is registered to SmartPlant Foundation.

Unregistered

If you access **Properties** on a single document and your model has not been registered to SmartPlant Foundation, the **Revision** tab displays previous entries made. A new row is available to make a new entry. You can edit each field using alphanumeric and special characters.

If you access **Properties** on a single component, multiple components, or multiple documents and your model has not been registered to SmartPlant Foundation, the **Revision** tab has a single blank row for a new or edited entry. All fields are editable. Their values are propagated to the writeable documents that are associated with the selected set.

Registered

If your model has been registered to SmartPlant Foundation, use the **Revise** command to create revision numbers. This command reserves a revision number by adding it to the document Revision properties. The revision number is added in the form of a blank row on the **Revision** tab of the **Properties** dialog box.

After reserving the revision number, right-click the document and select **Properties**. Go to the **Revision** tab and edit the **Revision** fields. All fields except for **Revision Mark** and **Revision Minor Number** are editable. For more information, see *Revising* (on page 421).

■ NOTES

- You can create more than one revision per instance of the **Properties** dialog box by selecting **Apply** after adding a record.
- You can delete one or more revision records by highlighting the revision rows and pressing Delete. You must select OK or Apply to make the deletion permanent. The rows selected for deletion must be adjacent and must include the last revision record.

See Also

Properties Dialog Box (on page 49)

WBS Tab (Properties Dialog Box)

Sets options for the Work Breakdown Structure (WBS) of drawings and reports. This tab is available only when your model has been registered using the SmartPlant Registration Wizard.

If you access this tab after selecting multiple drawings, these fields appear empty, regardless of the information defined for any of the drawings individually. Any information you add to this tab overrides the previously defined information in corresponding rows in the selected drawings.

Name

Displays the name of the property.

Value

Sets the current value of the property.

Behavior

Specifies whether to inherit or override a property in the hierarchy of items in the **Console**. If the property is **Inherited**, the setting comes from items higher in the hierarchy. If the property is not read-only, you can provide a new, overriding value for the property by changing the behavior setting to **Override**. This new value then propagates to other items deeper in the hierarchy. If you set the behavior to **Force Override**, the property setting is forced to items deeper in the hierarchy, even if they are set to **Inherit**.

Properties

Project Name

Displays the project to which the item belongs. In SmartPlant Foundation, a project is the scope of work approved for capital expenditure (that is, a job).

Document Type

Specifies the type of document, such as Civil Plan.

Document Style

Specifies the style of document, such as Ortho for orthographic drawing.

Discipline

Specifies the discipline for the document. If this is a 3D Model Data document, set the property to **SmartPlant Review Document**. If it is a drawing or report document, set the discipline to match the type of document.

Allow Publish

Sets the document as a publishable document.

Working with the Integrated Environment

You can only publish documents after the appropriate properties are set on the **WBS** tab. The **WBS** tab is not available if your login is not authenticated as valid for SmartPlant Foundation. The properties that must be defined for publishing are: **Document Type**, **Document Style**, **Discipline**, and **Allow Publish**. For more information, see *Set properties for publishing documents* (on page 428).

See Also

Properties Dialog Box (on page 49)

Configuration Tab (Properties Dialog Box)

Displays the creation, modification, and status information for an item.

Model

Displays the name of the Model. This value is read-only.

Permission group

Specifies the permission group to which the item belongs. You can select another permission group if needed.

If you access this tab after selecting multiple drawings, this field will be empty, regardless of what information was defined for any of the drawings individually. Any selection you in this field will overwrite the previously defined information in the selected drawings.

Transfer

Reassigns ownership of the selected model objects from their current permission group to another satellite or host permission group. This option is only available if the active model or project is replicated in a workshare configuration. The option is not available if all of the objects in the select set already belong to another location and are non-transferable. For more information, see *Transfer Ownership Dialog Box* in the *Common User's Guide*.

NOTE The **Transfer** option does not apply to the filters and surface style rules.

Approval State

Shows the approval state of the selected item. This value is read-only.

Status

Specifies the current status of the selected **Console** hierarchy item or items or selected documents in the **Detail View**. Depending on your access level, you may not be able to change the status of the selected items.

Date Created

Displays the date and time the item was created.

Created by

Displays the name of the person who created the item.

Date Last Modified

Displays the date and time the item was modified last.

Last Modified by

Displays the name of the person who modified the item last.

See Also

Properties Dialog Box (on page 49)

Transfer Ownership Dialog Box

Allows you to specify a new location and permission group for the selected model objects.

Current location

Displays the name of the location with which the current permission group is associated. All of the objects in the select set must belong to the same location.

Current permission group

Displays the name of the permission group with which the selected objects are currently associated. If all of the objects in the select set do not belong to the same permission group, this box appears blank.

New location

Specifies the name of the location to which you want to assign the objects. In a global workshare configuration, this box lists all the locations in which you have write access to one or more permission groups. The selection in this box filters the entries in the **New permission group** box.

New permission group

Specifies the new permission group to which to assign the selected objects. If you specify a value in the **New location** box, this list displays all permission groups to which you have write access in the selected location. If you do not specify a value in the **New location** box, this list includes all permission groups to which you have write access in all locations except the current location. This box is blank if you do not have write access to any permission groups at any locations other than the current one.

NOTE We strongly recommend that administrators follow naming convention rules that include the location as a prefix in the permission group name.

Choose Label Dialog Box

Specifies a label for a document property. This dialog box displays the labels available on the application server in the [Reference Data Folder]\SharedContent\Labels\Base Templates folders.

See Also

Setting Properties (on page 47)

Rename Command

Allows you to type a new name for a selected item in the hierarchy. The shortcut key for this command is **F2**.

NOTE You cannot have duplicate names at the same level in the tree, but the names are case-sensitive. For example, you can have two items be named 'ItemName' and 'itemname' at the same level in the tree.

Rename an item

- 1. Right-click an item in the **Management Console** hierarchy or **Detail View**, and select **Rename** on the shortcut menu. Alternatively, press **F2** on the keyboard.
- 2. Type a new name for the item.

See Also

Rename Command (on page 69) Shortcut Menus (on page 33) Components Overview (on page 89)

Run Query (Shortcut Menu)

Runs the query associated with the selected Drawings by Query Manager component. For example, if you execute **Run Query** on an Isogen Isometric Drawing by Query package associated with a Drawings by Query Manager, the software looks for piping in the model. The query results display beneath the style in the **Management Console**. You can create isometric drawings from the query results by right-clicking on the component and selecting **Create Drawing(s)**.

If you have an Orthographic Drawing by Query package associated to a Drawings by Query Manager, the **Run Query** command runs the query associated with the components, collects the objects from the database, and builds the information that will be included in the drawing. You can then create orthographic drawings from the query results by running the **Create Drawing(s)** command on the Orthographic Drawing component.

The Run Query command uses the filters specified when you performed Setup components.

■ NOTES

- When working in a Global Workshare Configuration with users logging into both Host and Satellite systems for more than one site, you can encounter problems with the filters defined for a Drawing by Query package. For example, if you create a Filter Root Folder for a particular site, and then you try to run queries for drawings associated to this filter from another site, the Run Query command is unable to update the drawings due to a lack of permission against the Filter Root Folder. You have to move or transfer the Filter Root Folder to the appropriate site where the Run Query command is executed.
- If the selected package was created and added manually, the folder name for the package must match the value for **pkgid** in the XML file. Edit the name of the folder to match the **pkgid** value.
- ★ IMPORTANT In marine mode, this command displays in the Ship Root and Folder shortcut menus for drawings by rule, but is not used with drawing by rule components.

See Also

Orthographic Drawings by Query (on page 213) Create Drawing(s) Command (on page 39)

Save As Command

Saves drawings and reports as specified file types to an external location, such as a share on another server. This command is not available until you generate drawings for at least one of the structures in the hierarchy. This command saves only the structures that contain drawings or

reports. You can save multiple file types based on the types of documents available. You can specify the target file type for each drawing type you want to save.

To save the hierarchy as a package, right-click the folder and select **Save Package** on the right-click menu. For more information, see *Save Package Command* (on page 76).

Save As Dialog Box (on page 75)

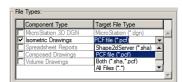
What do you want to do?

- Save to a file (on page 71)
- Retrieve piping component file data (PCF) (on page 72)
- Save as SmartPlant Foundation (*.xml) (on page 73)

Save to a file

Prior to following this procedure, you must have generated drawings already for at least one of the items in the **Console** hierarchy. The **Save As** command is not available if drawings have not been generated.

- 1. Select a folder, component, or document. You can select multiple documents by holding **Ctrl** or **Shift** and clicking each item.
- 2. Right-click your selection, then select Save As on the shortcut menu.
 - The Save As Dialog Box (on page 75) displays.
- 3. Specify the **Output Folder Rule** to be used. You can save the item as it displays in the hierarchy, with its parent folder appended or with the entire model hierarchy appended.
- 4. Specify the **File Already Exists Action** to be used. This determines how you save the file if it has the same name of an existing file. Select **Overwrite** to replace the existing file, or select **Save As filename (n)** to save the file separately.
- 5. Specify the **Output Folder** location. Click **Browse** to display a dialog box to locate the appropriate folder location.
- 6. Check the boxes for the **Component Types** you want to save. You can select multiple component types. For more information, see *Save As Dialog Box* (on page 75).
- 7. In the Target File Type lists, specify the file types you want to save. You can specify a file type for each component type selected. For example, you could use the Iso_Stress style to create a Piping Component File (PCF) file, then when you perform a Save As on the document, check the Isometric Drawings component type and specify the Target File Type as PCF File.



■ NOTES

 For Isogen isometric drawings, the name of the drawing document becomes the prefix for all of the files. For example, if the drawing document name is My_Pipeline, the saved file names become: My_Pipeline.sha, My_Pipeline.pcf, and so on.

- For Isogen isometric drawings, a file is created for each sheet in the drawing with [drawing name]_[sheet name] as the filename. For example, if the drawing My_Pipeline contains Sheet1 and Sheet2, two files will be saved with the names My_Pipeline_Sheet1 and My_Pipeline_Sheet2.
- If you configured a **StartProgram**, then the extracted PCF file will be stored with the relation <*Filename*>-extracted-pcf. For example, if the final PCF file name is My_Pipeline, then the extracted PCF is saved as My_Pipeline-extracted-pcf in the same location along with the My_Pipeline file.
 - ★IMPORTANT You must configure a **StartProgram** to store the extracted PCF. Otherwise, the software considers the final PCF as the extracted PCF.
- The SmartPlant Foundation (*.xml) target file type allows you to save drawing data and metadata to generate .xml files in either integrated and non-integrated environments that can be easily updated. For integration, you can use SmartPlant Interop Publisher to translate the generated .xml data file to a Smart Drawing available in SmartPlant Foundation. This option is available for Isometric Drawings, Composed Drawings, and Volume Drawings component types. For more information, see Save as SmartPlant Foundation (*.xml) (on page 73).
- 8. Click **OK** to save the files as specified.

■ NOTES

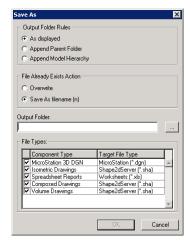
- The saved drawings retain the same names they had in this task.
- You can also extract all the sheets of a document to foreign formats such as MicroStation DGN, AutoCAD DWG and DXF files. A file is created for each sheet in the drawing with [drawing name]_[sheet name] as the filename. For example, if the drawing Volume_Drawing contains Sheet1 and Sheet2, two files are saved with the names Volume_Drawing_Sheet1 and Volume_Drawing_Sheet2.

Retrieve piping component file data (PCF)

When you create a Piping isometric document that uses the Iso_Stress style, the software does not create drawings. Instead it creates the data necessary for a Piping Component File (PCF). You can then output the PCF to the CAESAR II pipe stress analysis software.

To retrieve the PCF data, you use the Save As command.

 Right-click the Piping Isometric document that uses the Iso_Stress style and select Save As. The Save As dialog box displays.



- Specify the Output Folder Rule to be used. You can save the item as it displays in the Console, with its parent folder appended or with the entire model hierarchy appended.
- Specify the File Already Exists Action to be used. This determines how you save the file if
 it has the same name of an existing file. Select Overwrite to replace the existing file, or
 select Save As filename (n) to save the file separately.
- 4. Specify the **Output Folder** location. Click **Browse** to display a dialog box to locate the appropriate folder location.
- 5. Check the **Isometric Drawings** component type. You can select multiple component types. For more information, see *Save As Dialog Box* (on page 75).
- 6. In the **Target File Type** list for the Isometric Drawing component type, select **PCF File** (.pcf).
- 7. Click **OK** to save the files as specified.

The PCF file is saved to the location you specified, ready for use in stress analysis.

■ NOTES

- The saved drawings retain the same names they had in this task.
- You can also view the PCF data with the View Extraction Data command. For more information, see View Isogen isometric extraction data.

See Also

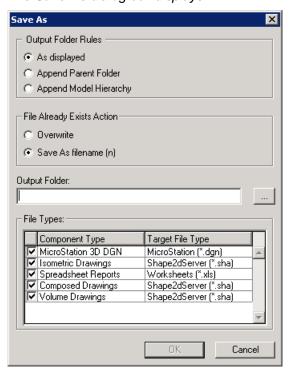
Save As Command (on page 70)

Save as SmartPlant Foundation (*.xml)

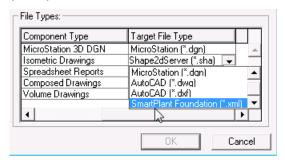
Use the **SmartPlant Foundation (*.xml)** target file type to save drawing data and metadata and generate .xml files in either integrated or non-integrated environments. This save option is available for **Isometric Drawings**, **Composed Drawings**, and **Volume Drawings** component types.

- Before you use the Save As command, you must define the Discipline property for your documents:
 - a. In the **Console**, right-click an item in the hierarchy, then select **Properties** on the shortcut menu.
 - b. Go to the WBS Tab.
 - c. Set the **Discipline** property. If your model has been registered using the SmartPlant Registration Wizard, this also adds the **Publish** command to the right-click menu for the selected document or documents. For a 3D Model Data document, the property is set to **SmartPlant Review Document**.
- 2. Right-click the Isometric, Volume, or Composed document, and select **Save As**.

The Save As dialog box displays.



- 3. Specify the **Output Folder Rule** to use. You can save the item as it displays in the **Console**, with its parent folder appended or with the entire model hierarchy appended.
- 4. Specify the File Already Exists Action to be used. This determines how you save the file if it has the same name of an existing file. Select Overwrite to replace the existing file, or select Save As filename (n) to save the file separately.
- 5. Specify the **Output Folder** location. Click **Browse** to display a dialog box to locate the appropriate folder location.
- 6. Check the **Isometric Drawings, Composed Drawings, or Volume Drawings** component type. You can select multiple component types.
- 7. In the **Target File Type** menu list for the component type, select **SmartPlant Foundation** (*.xml).



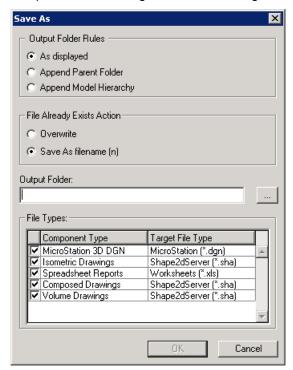
8. Click **OK** to save the files as specified.

The software generates an .sha file and two XML files in the specified location; an .xml data file and an .xml metadata file. The data .xml file is named ToolData_<Drawing Name>. The metadata file is named MetaData_<Drawing Name>.

You can now add the generated .sha in SmartPlant Interop Publisher to translate it to a Smart Drawing and publish the drawing to SmartPlant Foundation. SmartPlant Interop Publisher requires all generated files (.sha and .xml) to be located in the same folder location for translation and publishing. For more information about Smart Drawings and the translation and publish capabilities, refer to the SmartPlant Interop Publisher User's Guide.

Save As Dialog Box

Sets options for exporting drawings. You can open this dialog box by right-clicking a folder, component, or drawing and then selecting **Save As** on the right-click menu.



Output Folder Rules

Specifies how you want to save the hierarchy. The following options are provided:

- As displayed specifies the selection is saved as displayed.
- Append Parent Folder specifies the selected item is appended to the parent folder.
- Append Model Hierarchy specifies the selected item is appended to the hierarchy starting from the root and including the selected item.

File Already Exists Action

Specifies how you want to save the hierarchy if a file with the same name already exists.

- Overwrite overwrites the existing file.
- Save As filename (n) saves the file separately, and appends a number 'n' after the file name. For example, if the existing file name is FileName, the new file name is FileName (1).

Output Folder

Specifies the location to which to save the package.

Browse

Indicates a folder in which to save the drawings. You can select a local folder or a folder on another computer on the network.

File Type

Specifies the file formats to save for each drawing type. Check the box next to each drawing type you want to save. Use the **Target File Type** menu to specify the file type to which the drawing type is saved. The file types available for each drawing type are described in the following table.

Drawing Type	Target File Types
MicroStation	MicroStation J (V7 and V8) (*.dgn) 32MB limit
Isogen Isometric Drawings	Shape2DServer (*.sha)
■ NOTES	PCF file (*.pcf)
■ For Isogen Isometric Drawings, the name of the drawing document becomes the prefix for all of the files. For example, if the drawing document name is <i>My_Pipeline</i> , the saved file names become: <i>My_Pipeline.sha</i> , <i>My_Pipeline.pcf</i> , and so on.	Both (*.sha & *.pcf)
	All Files (*.*) - Includes all .sha and .pcf files, as well as all enabled supplementary files.
For Isogen Isometric Drawings, a file is created for each sheet in the drawing with [drawing name]_[sheet name] as the filename. For example, if the drawing My_Pipeline contains Sheet1 and Sheet2, two files will be saved with the names My_Pipeline_Sheet1 and My_Pipeline_Sheet2.	MicroStation J (V7 and V8) (*.dgn)
	AutoCAD (*.dxf)
	AutoCAD (*.dwg)
	SmartPlant Foundation (*.xml)
Spreadsheet Reports	Worksheets (*.xls)
Composed Drawings, Volume Drawings, and Orthographic Drawings by Query	Shape2DServer (*.sha)
	MicroStation J (V7 and V8) (*.dgn)
	AutoCAD (*.dxf)
	AutoCAD (*.dwg)
Composed by Drawings and Volume Drawings	SmartPlant Foundation (*.xml)

See Also

Save As Command (on page 70)

Save Package Command

Saves the **Management Console** hierarchy from the selected component down. The package saves the setup information and any template definitions that exist on nodes within the selected hierarchy. You can access this command by right-clicking a folder or application component in the **Management Console**. You must have at least write permissions on the component to access the **Save Package** command.

★ IMPORTANT This command is not used in the normal workflow of creating drawings and reports. An administrator with write permissions to the SharedContent folders saves packages.

When a package is added back into the **Management Console**, it recreates the hierarchy that was saved with the package.

Packages are also used in the setup of a Drawings by Query Manager component for the creation of orthographic and Isogen isometric drawings.

Save Package Dialog Box (on page 77)

■ NOTES

- Output documents are not saved in a package.
- If the topmost component saved in the package is a folder, then the package can be placed under the model root or a folder. If the topmost component saved in the package is an application component, then the package can only be placed under a folder.
- When you place a package, the software adds all the components to the active permission group.
- To save drawings or reports externally, see Save As Command (on page 70).

Save a package

- Right-click a component in the Management Console hierarchy, and select Save Package
 The Save Package dialog box displays.
- 2. Specify a package name, package description, and tab name. For example, you can save a package named **Piping Isometric Drawing Package** to a new tab called **Iso**.
- 3. Click **OK** to save the package.

The package is added to the **Add Component** dialog box on the specified tab. If a new tab name was specified, a new tab is added to the dialog box.

■ NOTE You can add the new package to the hierarchy by using the **New** command. Select the package on the **Add Component** dialog box. When you add a package, the software adds all the components to the active permission group.

See Also

Save Package Command (on page 76) Shortcut Menus (on page 33) Components Overview (on page 89) Orthographic Drawings by Query Common Tasks (on page 214)

Save Package Dialog Box

Package Name

Specifies a name for the package.

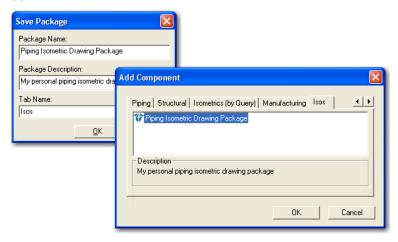
Package Description

Describes the package.

Tab Name

Specifies the tab of the **Add Component** dialog box on which the package displays. You can pick an existing name or type a new tab name in this field. The next time you access the

Add Component dialog box from an existing folder, the software adds the new tab and lists the new package on the tab. For more information, see *Add Component Dialog Box* (on page 43). For example, if you saved a package called Piping Isometric Drawing Package and added it to a new tab called Isos, an **Isos** tab is added to the **Add Component** dialog box:



See Also

Shortcut Menus (on page 33) Save a package (on page 77)

Setup Command

The **Setup** command sets options for a component or package. Each type of component has a unique **Setup** dialog box.

Setup (3D Model Data Component) (on page 190)

Setup (Generic Module Folder Component) (on page 198)

Setup (Drawings by Query Manager Component Shortcut Menu) (on page 219)

Setup (Drawings by Rule Component) (on page 104) (Marine mode only)

Setup (Orthographic Drawing by Query Component) (on page 217)

Setup (MicroStation DGN Files) (on page 194)

Setup (Search Folder) (on page 234)

Setup (Imported Folder) (on page 239)

Updating Documents

Updating documents increases productivity because you can easily keep deliverables current. It is important to understand the different update capabilities.

■ NOTES

- You must install the SmartPlant Schema Component to update documents.
- If the drawing document you are looking at in the **Detail View** has a yellow icon (for example: (a), the drawing document is a version 6.1 legacy Snapshot drawing. You should use the **Tools > Convert Legacy Snapshots** command to convert this document to a

- Composed Drawing for use in the current version of the software. If you do not convert the legacy snapshot drawing, you cannot update, revise, or publish the drawing.
- Any time you update a 3D model document, the software generates a single log file listing status information and any errors encountered during the process. The log file location is %TEMP%\EFUpdateCache\[3D Model ID]\[3D Model ID]\log.

Refreshing Document Contents

The **Refresh** command on the shortcut menu for a Console hierarchy item allows you to see which documents are out-of-date. For more information, see *Refresh* (*Shortcut Menu*) (on page 79).

Updating Documents Using Batch Processing

The **Batch** command queries the model to regenerate a single document or multiple documents. For volume and composed drawings, if you have edited the previous copy of the drawing (for example, by moving a label), the software remembers those changes and re-creates them. If batch processing is configured for the selected drawings, the update is performed on the Batch Server.

For more information, see Batch Processing - Intergraph Smart Batch Services (on page 241).

See Also

Refresh document status (on page 80) Print Command (on page 45)

Refresh (Shortcut Menu)

Compares the date of the last update of the document with the modification date in the model for any object that has a *positive* (can be seen) resymbolization in the drawing.

This command is not available until you generate documents. You can access this command by right-clicking an item in the **Management Console** hierarchy or in the **Detail View** and selecting **Refresh** on the shortcut menu. The software updates the status for all the expanded items within the parent node.

■ NOTE The Refresh command is not available for Spreadsheet Report documents. A Spreadsheet Report document is regenerated each time you update or print the report document. You can refresh the contents of a folder that contains reports.

For out-of-date documents X, the command behavior implies the following:

- If the object is hidden entirely and is inside the drawing volume, but the style does not resymbolize the hidden lines, the object does not participate in the "out-of-date" definition.
- The modification date used for the object in the model can be for any property even if this property has no impact on the graphic. This means that a drawing could be considered out-of-date even though the graphic is up-to-date. For example, the approval status does not affect graphics. However, the view style you are using for your drawing can use a filter that sets approved objects to a specific color. A drawing document displays with an out-of-date definition because of a change to the approval status.

 Objects participating indirectly in the graphic as labels do not participate in the out-of-date definition. This means that, in rare cases, a label may be out-of-date on a drawing that is shown as up-to-date.

■ NOTES

- To refresh the Management Console to reflect changes made to loaded (expanded) tree view items, use the View > Refresh command.
- You can also use the Batch > Refresh command on the shortcut menu to perform your refreshes on the batch server. Batch > Refresh is not available for Spreadsheet Report documents.

Refresh document status

- 1. Right-click any item in the Management Console or in the Detail View.
- 2. On the shortcut menu, click **Refresh**. The software checks the model for any differences. The drawing icons change to reflect the status of the documents compared to the model.

■ NOTE After refreshing a folder or component, you can synchronize it with the model by right-clicking the item and then selecting **Update** (drawings by rule) or **Update Now** on the shortcut menu.

See Also

Refresh (Shortcut Menu) (on page 79) Icons for Components and Drawings (on page 25) Updating Documents (on page 78)

Update Now

Updates a single document or multiple drawing documents whether or not they are out-of-date. This command is available when you right-click on:

- A folder or component in the Management Console. All drawings in all components within the folder are updated.
- A component in the Management Console. All drawings in the component are updated.
- One or more documents in the **Detail View**. The selected drawings are updated.

This command works on your local computer regardless of the batch configuration.

For volume drawings, the **Update Now** command is not available until you place drawing volumes for a volume component in the Space Management task. For composed drawings, this command is not available until you create the drawings in a 3D task through **Tools** > **Drawing Console**. For reports, this command is not available until you create the report by choosing a report template.

■ NOTES

- If the software encounters a problem before or during the drawing update, it stops updating, displays either an error status or error message, and saves the errors to the log file. For more information, see Conditional Drawing Update (on page 86).
- The software preserves many of the modifications you make between regenerations of volume drawings. For example, if you annotate a volume drawing and then regenerate it, your annotations still display on the updated drawing.

- Documents created automatically in a Drawings by Rule component are automatically deleted by updating the component if the document no longer contains views.
- If the software cannot make a SmartPlant Foundation server connection when updating 3D Model Data documents, you are prompted to provide a valid login and password.
- If you place drawing property labels on a template, generate a drawing, move the labels on the drawing, and then update the drawing, the software remembers the new position of the labels on the drawing.

What do you want to do?

- Update all drawings in a folder or a component (on page 81)
- Update a report (on page 81)
- Update a single drawing (on page 82)
- Insert a note at a precise place on an isometric drawing (on page 82)

Update all drawings in a folder or a component

- NOTE Before you update a folder or a component, you can refresh its documents to determine which documents are out-of-date. For more information, see *Refresh document status* (on page 80). You do not have to refresh before updating, but it can be helpful to determine which documents are out-of-date.
- 1. Right-click a folder or a component in the hierarchy. If you select a component, it must contain existing drawings or reports. If you select a folder, it must contain a component with existing drawings or reports.
- 2. On the shortcut menu, click **Update Now**.

The icons for the out-of-date documents change to show they are updated. If the Batch Server is configured, the command displays the **Drawing Batch** dialog box.

For more information, see *Batch Processing - Intergraph Smart Batch Services* (on page 241).

■ NOTE You can update an individual document by right-clicking the document and selecting **Update Now** on the shortcut menu.

Update a report

- 1. Right-click a report document.
- On the shortcut menu, select Batch > Update to update the document on the batch server now or create a schedule to run the batch job. Alternatively, select Update Now to update locally.

The icon for the out-of-date document changes to show it is updated \checkmark .

3. If the batch server is configured, the **Drawings Batch** dialog box displays.

Batch Processing - Intergraph Smart Batch Services (on page 241)

■ NOTE When using the **Batch** > **Update** command and batch processing is configured for the selected document, the update is performed on the Batch Server. If batch processing is not

configured, the command behaves the same as the **Update Now** command, performing a complete regeneration of the entire drawing on the local machine.

Update a single drawing

- 1. Right-click a document.
- On the shortcut menu, select Batch > Update to update the document on the batch server now or create a schedule to run the batch job. Alternatively, select Update Now to update locally.

The icon for the out-of-date document changes to show it is updated ✓.

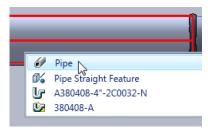
3. If the batch server is configured, the **Drawings Batch** dialog box displays.

Batch Processing - Intergraph Smart Batch Services (on page 241)

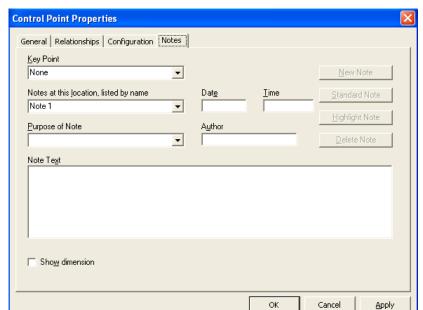
NOTE When using the **Batch** > **Update** command and batch processing is configured for the selected document, the update is performed on the Batch Server. If batch processing is not configured, the command behaves the same as the **Update Now** command, performing a complete regeneration of the entire drawing on the local machine.

Insert a note at a precise place on an isometric drawing

- 1. Select Insert > Control Point.
- 2. Select the pipe part in the model that needs a note.
 - TIP Use the quick pick tool to make the correct selection.



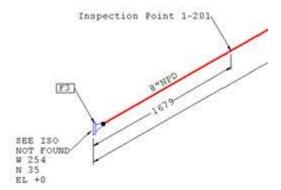
- 3. Position the control point on the centerline of the pipe part. For precision placement, use commands like **Measure** , **PinPoint** , or **Point Along**.
- 4. In the **Type** box, verify that **Control Point** is selected.
- 5. Confirm or change the option in the **Subtype** box on the ribbon.
- 6. In the Name box, define a name for the control point.
- Click Properties 2.
- 8. Under the **Notes** tab, click **New Note**.
- 9. In the **Key point** box, select the control point to which to attach the note.
- 10. In the **Notes at this location, listed by name** box, type or select a name.
- 11. In the **Purpose of note** box, select **Fabrication** so that the note is picked up for inclusion in the drawing.
- 12. In the **Note text** box, type descriptive text for the note.



13. Check the Show dimension box if you want a dimension to appear on the drawing.

- 14. Click **OK**.
- 15. In the Drawings and Reports task, use the **Update Now** or **Batch** > **Update** commands to update the drawing that includes the pipe part with the associated control point.

The note appears where it was placed in the model.



Update

In marine mode or material handling mode, **Update** performs two functions:

- For existing drawings, this command updates a single document or multiple drawing documents.
- When drawings have not been created, this command generates and updates the drawings for the selected component. This option is only available for drawings-by-rule components.
 - NOTE Marine mode or material handling mode only: If you did not previously select the model contents for a drawings-by-rule component, the Filter Properties for Asking Filter

dialog box displays, and you must select a filter. The dialog is the same as the **Filter Properties** dialog box, except that only the appropriate tabs for the asking filter are displayed. For more information, see *Filter Properties Dialog Box* (on page 164).

For other plant mode drawings, this command updates a single document or multiple drawing documents.

Update is available when you right-click on:

- A folder or component in the Management Console. All drawings in all components within the folder are updated.
- A component in the Management Console. All drawings in the component are updated.
- One or more documents in the **Detail View**. The selected drawings are updated.

This command works on your local computer regardless of the batch configuration.

Update performs a smart update of a drawing. If a set of criteria is met, then an incremental update is performed only for the geometry of added, modified, and deleted objects. If the criteria are not met, then a full update of all geometry is performed.

All of the following criteria must be met for **Update** to trigger an incremental update:

- Each view in the drawing has had at least one full update previously performed.
- The only view properties that have changed are name and description. For more information, see View Tab (Drawing View Properties Dialog Box) (on page 277).
- The numbers of added, modified, or deleted objects in each view are less than a set percentage of the total number of objects.
- The software allows incremental updates for the view types in the drawing.

Any one of the following criteria cause **Update** to trigger a full update:

- A drawing view is new and has never had a full update.
- A drawing view is a section or detail view.
- A drawing view has been cropped.
- The view style, scale, orientation, or coordinate system of a drawing view have changed.
- A primary plate object in a view has been spit or unsplit.
- The numbers of added, modified, or deleted objects in each view are greater than a set percentage of the total number of objects.
- The software does not allow incremental updates for the view types in the drawing.

The **Update** command is not available until you create drawings in the **Management Console** using the **Create Drawing(s)** command. When a folder or component is selected, all views in the component(s) are updated, including unassigned views in the **Unassigned Folder** of the Drawing View Explorer.

■ NOTES

- If you have not previously selected a filter for the component, the Select Filter dialog box displays, and you must select a filter. For more information, see Select Filter Dialog Box (on page 122).
- An incremental update is usually faster than a full update, but still results in completely up-to-date geometry for the view.

- Documents created automatically in a Drawings by Rule component are automatically deleted by updating the component if the document no longer contains views.
- The Update command is only available for drawings by rule. For other drawing types, the Update Now command is used. For more information, see Update Now (on page 80).

What do you want to do?

- Update all drawings in a folder or a component (on page 85)
- Update a single drawing (on page 85)
- Create automated major views for steel order scantling drawings (on page 154)
- Create an assembly drawing (on page 131)

Update all drawings in a folder or a component

■ NOTE Before you update a folder or a component, you can refresh its documents to determine which documents are out-of-date. For more information, see *Refresh document status* (on page 80). You do not have to refresh before updating, but it can be helpful to determine which documents are out-of-date.

- Right-click a drawings-by-rule folder or component in the hierarchy. The component must contain existing drawings or reports. The folder must contain a component with existing drawings or reports.
- On the shortcut menu, click **Update**. The icons for the out-of-date documents change to show they are updated. If the Batch Server is configured, the command displays the **Schedule Wizard**.

Batch Schedule Wizard Common Tasks

■ NOTES

- Documents created automatically in a Drawings by Rule component are automatically deleted by updating the component if the document no longer contains views.
- You can update an individual document by right-clicking the document and selecting Update or Batch > Update on the shortcut menu.
- If you place drawing property labels on a template, generate a drawing, move the labels on the drawing, and then update the drawing, the software remembers the new position of the labels on the drawing.

Update a single drawing

- 1. Right-click a drawings-by-rule document.
- On the shortcut menu, select Batch > Update to update the document on the batch server
 now or create a schedule to run the batch job. Select Update to update locally. The icon for
 the out-of-date document changes to show it is updated .
- 3. If the batch server is configured, the **Schedule Wizard** displays.

Batch Schedule Wizard Common Tasks

■ NOTES

- When using the Batch > Update command and batch processing is configured for the selected document, the update is performed on the Batch Server. If batch processing is not configured, the command behaves the same as the Update command, performing a complete regeneration of the entire drawing on the local machine.
- If you place drawing property labels on a template, generate a drawing, move the labels on the drawing, and then update the drawing, the software remembers the new position of the labels on the drawing.

Conditional Drawing Update

Before updating a drawing, Smart 3D ensures necessary information is present before changing the existing drawing document. If any of the following conditions are true before the update process begins, the software displays an error message that lists the missing items, and makes no changes to the drawing. If any of the following conditions are true after the update process begins, the drawing update process stops, changes to an error status (), and Smart 3D preserves the previous state of the drawing document to avoid any data loss.

Reference Data Conditions

- View style cannot be found.
 - Graphic preparation rule cannot be found.
 - Graphic rule cannot be found.
 - Label rule cannot be found.
 - Dimension rule cannot be found.
 - North arrow rule cannot be found.
 - Matchline rule cannot be found.
 - View rule cannot be found.
- View style filter cannot be found.
- Emptyvw.sha file cannot be found.
- Styles.sha file cannot be found.

Model Data Conditions

- Associated volume cannot be found.
- Associated coordinate system cannot be found.

Memory Conditions

- Not enough available memory to begin the update process.
- Not enough available memory to complete the update process after it has started.

■ NOTES

Failure conditions found before the update process can be viewed in the **Drawings.log**.

- Failure conditions found during the update process can be viewed when you right-click a drawing and select View Log.
- Failure conditions are not logged when the **Update** command is used on a drawing view. You must use the **Update Now** command in either the **Drawing Console** or Drawings and Reports task to save the error(s) to the log.
- If an orphan view (a view that exists in the model database without a corresponding view) exists, it is removed from the database and a description of the error is logged in the Drawings log file in <Temp Folder>\Logs.

Restore

Restores a drawing document from a model restored from a backup. This command is available when you right-click on a document in the **Drawing Console** or the **Drawings and Reports** task.

■ NOTES

- When a document is restored, all document properties are overwritten.
- You cannot restore a document if the document is created in your current model after the backup is created. If you select a group of documents to restore, new documents in the group are not restored. All other documents are restored from the backup model.
- Views that are created in your current drawing after the backup is created are lost when you select Restore.

What do you want to do?

- Create a backup to use for restoration (on page 87)
- Restore a document from a backup model (on page 87)

Create a backup to use for restoration

- In the Project Management environment, create a backup file of your current model using Tools > Backup. For more information, see Backup in the Project Management User's Guide.
- 2. Create a new model in the site by restoring the backup file using **Tools** > **Restore**. For more information, see *Restore* and *Restore Wizard* in the *Project Management User's Guide*.
- TIP Give the new model a name that makes it easy to identify as the backup of your current model.

Restore a document from a backup model

- 1. In the **Drawing Console** or the **Drawings and Reports** task, right-click on a drawing document in your current model, and select **Restore**.
 - The Restore Document dialog box displays.
- 2. Select the backup model from the **Model** box, and then click **OK**.
 - The document in your current model is replaced with the document from the backup model.

View Log Command

Displays the log information for the selected drawing. To access this command:

Right-click a drawing document at any time, and select View Log on the shortcut menu.

For drawings by rule, right-click a component after using Create Drawing(s) or Update at the component level, and select View Log on the shortcut menu. View Log is only available if there was an error during the create or update operations.

SECTION 6

Components Overview

Several specialized components allow you to access commands for configuring templates and generating drawings and reports. The various types of components can be divided into two groups: application components and folder components. Many of the application components correspond to specific types of drawings, such as volume drawings and composed drawings. The Spreadsheet Reports component provides access to report-related commands. You organize drawings and reports in folders. You can add folders to the root and to other folders. Each component has a different icon and right-click menu.

You can copy and paste components with some restrictions. For example, application components cannot contain folders or other application components. If you copy a folder, you can paste it under a folder but not under another component type. If you copy a Volume Drawing component, you can paste it under a folder. If you copy other types of components, you can paste them under folders but not under other types of components.

Your administrator can assign permissions to the different components using commands in the Project Management task. For example, the administrator can set permissions so that only the piping designers have write privileges on Isogen Isometric Drawings. For more information, see the *Project Management User's Guide*.

There are several types of delivered components. Their names reflect the type of drawing or report they create. When you right-click the root or a folder and select **New**, the **Add Component** dialog box displays. The dialog box includes a **General Tab** for general types of drawings or reports and additional task-specific tabs with delivered folders and packages. For more information, see *Add Component Dialog Box* (on page 43).

■ NOTE You can also create and manage components from the 3D modeling tasks by using the **Tools > Drawing Console** command. For more information, see the *Common User's Guide*.

The following components are shown on the **General** tab of the **Add Component** dialog box. Other components are shown on other tabs.

Delivered Component

3D Model By Query

Description

Creates a 3D Model By Query component in the Console. You can use the 3D Model By Query component to export 3D model data in bulk as CAD (SAT) files. The software uses a filter-based query to collect the objects and document them automatically. More information is available in the Orthographic Drawings User's Guide.

Delivered Component

Description

3D Model Data

Creates a 3D Model Data component in the Console. Right-click the component and select **Setup** on the shortcut menu to specify a filter that identifies the objects you want collected by the 3D Model Data component. You can use the 3D Model Data component to output SmartPlant Review files or CAD (SAT) files. More information is available in the Orthographic Drawings User's Guide.

MicroStation 3D DGN

Creates a MicroStation 3D DGN component in the Console. Right-click the component and select **Setup** from the shortcut menu to define the MicroStation seed file and style to use in generating the component documents. More information is available in the Orthographic Drawings User's Guide.

Folder

Creates an empty folder in the Console.

Drawings By Query Manager

Creates a Drawings by Query Manager in the Console. The Drawings by Query Manager is used in conjunction with other components, such as the Orthographic Drawing by Query and Isogen Isometric Drawing by Query components, to complete the query for objects in the model. The Drawings by Query Manager provides the filter that specifies the "where" side of the query. It tells the query "where" to look for the objects specified by the component "what" filter. More information is available in the Orthographic Drawings Users Guide and the Isogen Isometric Drawings User's Guide.

Generic Module Folder

Creates a Generic Module Folder in the Console. The Generic Module Folder component provides a way for you to run your custom VB modules to create custom drawings. You set up the Generic Module Folder component to use your custom VB module. More information is available in the Orthographic Drawings User's Guide.

Imported Folder

Imports an external Windows folder from a shared network folder into the Management or Drawing console. The imported folder can contain any type of file available in Windows. You can manage the imported documents, and publish them using Imported Folder menu options. For more information, refer to Drawings Folders in the Orthographic Drawings User's Guide.

Delivered Component

Description

Composed Drawings

Creates a Composed Drawing component in the Console. Composed drawings are orthographic drawings created in a 3D task such as Common. The composed drawing component manages the composed drawings you create. More information is available in the *Orthographic Drawings User's Guide*.

Orthographic Drawings by Query

Creates an Orthographic Drawings by Query component in the Console. The Orthographic Drawing by Query component allows you to create drawings for many objects in the model all in the same manner. This component does not require physical volumes in the model. The software uses a filter-based query to collect the objects and document them automatically. More information is available in the *Orthographic Drawings User's Guide*.

Isogen Isometric Drawings by Query

Creates an Isogen Isometric Drawings by Query component in the Console. You create an isometric drawing by associating an Isogen Isometric Drawing by Query component to a Query Manager. The Isogen Isometric Drawing by Query component specifies the "what" portion of the query, while the Query Manager specifies the "where." More information is available in the *Isogen Isometric Drawings User's Guide*.

Spreadsheet Reports

Creates a Spreadsheet Report component in the Console. More information is available in the *Reports User's Guide*.

Search Folder

Creates a Search Folder in the Console. Search Folders allow you to search for documents based on common properties such as out-of-date status, approval, or documents that have been published to a certain contract in integrated environment. More information is available in the *Orthographic Drawings User's Guide*.

5 Volume Drawings

Creates a Volume Drawings component in the Console. The Volume Drawing component uses a template to create drawings. You can place a view on the template and associate the view with a view style to control the output. To define the contents of the view, you create a drawing volume in the model. More information is available in the *Orthographic Drawings User's Guide*.

For drawing components used in marine mode, see *Drawings by Rule Components* (on page 92) Drawings by Rule Components in the Orthographic Drawings User's Guide. You can access this document using the **Help > Printable Guides** command in the software.

See Also

Shortcut Menus (on page 33)
Icons for Components and Drawings (on page 25)
New Command (on page 41)
Delivered Drawing Types (on page 21)
Drawings and Reports Naming Rules (on page 95)
Document and Sheet Naming Rules in Drawings by Rule (on page 96)

Drawings by Rule Components

The delivered marine mode drawings by rule packages include the following:

On the **Ship** tab of the **Add Component** dialog box:

 Drawings by Rule - Creates a Drawing by Rule component in the Console. The component allows you to create empty drawings by rule to which you add the needed views.

Assembly Drawings

On the **Ship** tab of the **Add Component** dialog box:

- Assembly Method Creates views of individual parts and an isometric of the assembly.
- Assembly Sequence Creates views showing the sequence of parts.
- Assembly Creates plan, elevation, and isometric views for an assembly. All views are on a single drawing sheet.
- Assembly (Generic) Creates plan, elevation, and isometric views for an assembly using the more efficient generic ruleset. All views are on a single drawing sheet.

For more information, see Assembly Drawings (on page 129) in the Orthographic Drawings User's Guide.

Hull Line Drawings

On the **Ship** tab of the **Add Component** dialog box:

- Hull Lines Creates body plan, plan, and profile views of the hull lines.
- Body Plan Creates body plan views of the hull lines.

For more information, see *Hull Lines Drawings* (on page 137) in the *Orthographic Drawings User's Guide*.

Manufacturing Drawings

On the **Manufacturing** tab of the **Add Component** dialog box:

- Mfg PinJig Contains an asking filter to select the pin jigs.
- Mfg Profile Sketch Contains an asking filter to select the profile.
- Mfg Profile Sketch (Multiple) Contains an asking filter to select the profiles.
- Mfg Template Set Contains an asking filter to select the template sets.

 Mfg Templateset (Full Scale) Package - Contains an asking filter to select the template sets. For more information, see Mfg Templateset (Full Scale) Package in the Drawings and Reports Reference Data Guide.

For more information, see *Manufacturing Drawings* (on page 144) in the *Orthographic Drawings User's Guide*.

Scantling Drawings

On the **Ship Structure** tab of the **Add Component** dialog box:

- Shell Expansion Creates unfolded port and starboard views of shell plate.
- Steel Order by Block or Assembly Creates transverse, longitudinal, deck shell expansion, and shell profile views for plate and profile parts within selected blocks or assemblies.
- Steel Order by Reference Planes Creates a single drawing for each reference plane selected as input.
- Steel Order (Expansion) Creates expanded shell profile views for plate and profile parts.
- Steel Order (Shell Profiles) Creates plan views of individual shell profiles, also showing level 2 non-target parts connected to the profiles.
- Steel Order by Block or Assembly (Generic) Creates transverse, longitudinal, deck shell expansion, and shell profile views for plate and profile parts within selected blocks or assemblies using the more efficient generic ruleset.
- Steel Order by Reference Planes (Generic) Creates a single drawing for each reference plane selected as input using the more efficient generic ruleset.
- Steel Order Shell Profiles (Generic) Creates plan views of individual shell profiles showing level 2 non-target parts connected to the profiles using the more efficient generic ruleset.

The Steel Order packages use scantling steel order rule sets and view styles for light (non-detailed) parts created in the Molded Forms task and detailed parts created in the Structural Detailing task. The packages also use short name and typical labels for profiles and plates.

For more information, see *Scantling Drawings* (on page 153) in the *Orthographic Drawings User's Guide*.

Offshore Drawings

On the **Offshore** tab of the **Add Component** dialog box:

- Member Parts Creates drawings of individual member parts and their connected parts.
 Two views are generated for each member part: one in the -Z direction and one in the +Y direction of the member local coordinate system.
- Member Parts (Generic) Creates drawings of individual member parts and their connected parts using the more efficient generic ruleset. Two views are generated for each member part: one in the -Z direction and one in the +Y direction of the member local coordinate system.
- Pipe Supports (Generic) Creates end, side, isometric, and detail views of a pipe support.
 All views are on a single drawing sheet.

• **Piping (Generic)** - Creates forward-looking elevation, port-looking elevation, isometric, and plan views for a pipeline. All views are on a single drawing sheet.

■ NOTE For volume-drawing-based orthographic piping packages, see *Piping Packages* in the *Drawings and Reports Reference Data Guide*.

Baseline Drawings by Rule Package

You can also use the baseline **Drawings by Rule** component to create an empty drawing component to which you can add views as needed. The **Drawings by Rule** component displays on the **Ship** tab of the **Add Component** dialog box. After adding the component to a Management Console folder, you can set it up with views and reports, specify the query definition, and create drawings. For more information, see *Drawings by Rule* (on page 102).

See Also

Components Overview (on page 89) Shortcut Menus (on page 33) New Command (on page 41) Icons for Components and Drawings (on page 25)

SECTION 7

Drawings and Reports Naming Rules

The software provides several options for naming drawing and reports. In addition to the delivered sample naming rules that are described below, you can create your own naming rules. For more information on creating naming rules, see the *Reference Data Guide*.

Default Drawing Name Rule

Creates a name based on the parent component name, Global Workshare location ID, and an index number. The naming rule inserts a "-" between each name. If there are no parent objects, then only the child object name is used. For example, the first drawing created under the ComposedDrawings001 component at workshare site 1 is called *ComposedDrawings001-1-0001*.

Default By Query Name Rule

Creates a name based on the parent object name (each system and sub-system above the child object will be included in the name) and child object name. The naming rule inserts a "-" between each name. If there are no parent objects, then only the child object name is used. For example, the first composed drawing created at workshare site 1 under the ComposedDrawings001 component would be named *ComposedDrawings001-0001*.

Default Report Name Rule

Creates a name based on the object name, Global Workshare location ID, and an index number. For example, the first drawing created at workshare site 1 is called *ComposedDrawings001-1-0001*.

Volume Name Rule

Creates a name based on the volume name, Global Workshare location ID, and an index number. For example, the first drawing created using the volume Volume001 in workshare site 1 is called *Volume001-1-0001*. If a volume is not specified, the drawing name will be named *Unspecified*.

User Defined

Allows you to define a custom name for the drawing. After selecting this naming rule, type the name for the drawing in the **Name** box.

See Also

Create a new composed drawing (on page 205) Create a volume drawing (on page 229) Delivered Drawing Types (on page 21)

Document and Sheet Naming Rules in Drawings by Rule

The software provides several options for naming drawings by rule documents and the individual sheets within the documents. For more information on creating naming rules, see the *Reference Data Guide*.

The following default naming rules are delivered:

Default

Document

Names the drawing using the following criteria:

<plant name><model name>"-"<Global Workshare Location ID>"-D"<index number>

For example, if a model named L101 has a location ID set to 12 in the Project Management task, the name of the second drawing created should be named **L101-12-00002**.

 Sheets - Names drawing sheets using the default software names. For example, two sheets are named Sheet1 and Sheet2.

Name by Assembly

Finds the first view on the drawing (as sorted internally by the software), finds the primary input object on the view (such as a system or part), and uses the parent assembly or block of the primary object, as defined by the Planning task.

Document

Names the drawing using the following criteria:

<model name>"-"<Global Workshare Location ID>"-"<assembly/block name>"-D"<index number>

For example, for a model name of L101, a location ID of 1, and an assembly name of A0123, the first drawing created based on the assembly name should be named **L101-1-A0123-D0001**.

Sheets

Names drawing sheets using the following criteria:

<assembly/block name>"_"<index number>

For example, three sheets using the same assembly name are named A0123, A0123_1, and A0123_2. Two sheets using different assembly names are A0123 and A4567.

When the criteria for the rule are not met, the software defaults to the **Default** rule.

Name by Query

Finds the first view on the drawing (as sorted internally by the software) and uses the primary input object on the view (such as a system or part). Also determines if the Global Workshare Location ID is part of the block/assembly name.

Document

Names the drawing using the following criteria:

<model name>"-"<Global Workshare Location ID>"-"<object name>"-D"<index number>

For example, for a model name of L101, a location ID of 1, and a plate part name of <B0.1>-LS.5-1, the second drawing created based on the part name should be named L101-1-<B0.1>-LS.5-1-D02.

Sheets

Names drawing sheets using the following criteria:

```
<object name>"_"<index number>
```

For example, three sheets using the same plate system name are named A-103DCK, A-103DCK_1, and A-103DCK_2. Two sheets using different assembly names are A-103DCK and A-110DCK.

When the criteria for the rule are not met, the software defaults to the **Default** rule.

Name by Grid Plane

Finds the first view on the drawing (as sorted internally by the software), finds the primary input object on the view, and looks for the parent system if the input object is not a system. The rule then uses the name of the grid plane that defines a plate system or the profile system name for a profile system. This rule is only used for planar plate systems or intersection profile systems.

Document

Names the drawing using the following criteria:

<model name>"-"<Global Workshare Location ID>"-"<grid plane name>"+ or -"<offset from grid plane>"-D"<index number>

If the drawing is for a shell profile, names the drawing using the following criteria:

<model name>"-"<Global Workshare Location ID>"- Shell Profile -"<parent profile system name>"-"<index number>

For example, for a model name of L101, a location ID of 1, and a plate system defined by a grid plane of F222, the third drawing created based on the grid plane should be named **L101-1-F222-D03**. For a grid plane of F222 and a global coordinate system offset of -0.5 m, the drawing is named **L101-1-F222-0.5-D03**. For a profile by intersection, the profile name is used, such as **L101-1-D47-1DCK-101T-D01**.

Sheets

Names drawing sheets using the following criteria:

```
<grid plane name>"+ or -"<offset from grid plane>" "<index number>
```

For example, three sheets using the same assembly name are named **F222**, **F222_1**, and **F222_2**. Two sheets using different assembly names are **F222** and **F232**.

When the criteria for the rule are not met, the software defaults to the **Default** rule.

Name by Asking Assembly

Document (automatically created, single view, shell plates)

If a block or assembly is selected through an asking filter, the drawing is named using the following criteria:

<model name>"-"<Global Workshare Location ID>"-SH-EX-"<assembly/block name>" - "<PORT/STBD>"-D"<index number>

For example, for a model name of L101, a location ID of 1, and a starboard block name of B0.1.2, the second drawing created based on the grid plane should be named **L101-1-SH-EX-BO.1.2-STBD-D03**.

If a single shell part is selected without the block/assembly asking filter, the drawing is named using the following criteria:

<model name>"-"<Global Workshare Location ID>"-SH-EX-"<part name>" "<PORT/STBD>"-D"<index number>

If multiple shell parts are selected without the block/assembly asking filter, the drawing is named using the following criteria:

<model name>"-"<Global Workshare Location ID>"-SH-EX-"<PORT/STBD>"-D"<index number>

Document (automatically created, single view, non-shell plates)

Names the drawing using the following criteria:

<model name>"-"<Global Workshare Location ID>"-"<grid plane name>"+ or -"<offset from grid plane>"-D"<index number>

For example, for a model name of L101, a location ID of 1, and a plate system defined by a grid plane of F222, the third drawing created based on the grid plane should be named **L101-1-F222-D03**. For a grid plane of F222 and a global coordinate system offset of -0.5 m, the drawing is named **L101-1-F222-0.5-D03**.

Document (automatically created, multiple views)

Names the drawing using the following criteria:

<model name>"-"<Global Workshare Location ID>"-MultipleViews-D"<index number>

For example, for a model name of L101, and a location ID of 1, the first drawing created based on the assembly name should be named **L101-1-MultipleViews-D0001**.

Document (manually created)

Use the **Add Document** command in the Management Console to manually create documents.

Names the drawing using the following criteria:

<model name>"-"<Global Workshare Location ID>"-"<assembly/block
name>"-D"<index number>

For example, for a model name of L101, a location ID of 1, and an assembly name of A0123, the first drawing created based on the assembly name should be named **L101-1-A0123-D0001**.

 Sheets - Names drawing sheets using the default software names. For example, two sheets are named Sheet1 and Sheet2.

When the criteria for the rule are not met, the software defaults to the Name by Query rule.

Name by Template Set Plate Part

This rule is applicable only to template set drawings.

Finds the first view on the drawing (as sorted internally by the software) and uses the primary input object on the view (such as a system or part). Also determines if the Global Workshare Location ID is part of the block/assembly name.

Document

Names the drawing using the following criteria:

<model name>"-"<Global Workshare Location ID>"-"<TemplateSet Plate Part
name>"-D"<index number>

For example, for a model name of L101, a location ID of 1, and a Templateset plate part name of <B0.1>-LS.5-1, the second drawing created based on the part name should be named L101-1-<B0.1>-LS.5-1-D02.

Sheets

Names drawing sheets using the following criteria:

< TemplateSet Plate name>"_"<index number>

For example, three sheets using the same plate system name are named A-103DCK, A-103DCK_1, and A-103DCK_2. Two sheets using different assembly names are A-103DCK and A-110DCK.

When the criteria for the rule are not met, the software defaults to the **Default** rule.

Name for Full Expansion

This rule is applicable only to full expansion drawings.

Finds the shell expansion view on the drawing and finds the hull object on the view. Also determines if the Global Workshare Location ID is part of the name, and whether a port or starboard view type has been selected.

Document

Names the drawing using the following criteria:

<model name>"-"<Global Workshare Location ID>"- SHELL EXPANSION -" <PORT or STBD> "-D"<index number>

For example, for a ship name of L101, a location ID of 1, and a view type of starboard, the drawing should be named L101-1- SHELL EXPANSION -STBD-D0001.

 Sheets - Names drawing sheets using the default software names. For example, two sheets are named Sheet1 and Sheet2.

When the criteria for the rule are not met, the software defaults to the **Default** rule.

See Also

Components Overview (on page 89)

View Naming Rules in Drawings by Rule

The software provides several options for naming the views in drawing by rule documents. For more information on creating view naming rules, see the *Reference Data Guide*.

The following view naming rules are delivered:

Default

Default naming convention is applied by the software. For example: View1.

Name by Block for Expansion

Uses the selected assembly, block, or assembly block name. This rule is used for shell expansions in a block or assembly.

Names the view using "SHELL EXPANSION - "<*PORT/STBD>*" - "<*assembly/block name>*. For example, for a starboard-side assembly name of A0123, the view is named **SHELL EXPANSION - STBD - A0123**.

Name by Direction

Finds the primary input part in the view and the view direction returned from the orientation rule.

Names the view using *<primary part name> "-" <view direction>*. For example, for a member part view, the name is **BU1900x750x15x50-1-001 - Along +Y**. For a view with a deck plate part as the primary input, the name is **<B412>-4LS.16-1 - Plan Looking Down**.

Name by Grid Plane

Finds the primary input plate part in the view and uses the plate type and the grid plane used to create the parent plate system of the part. This rule is used for plate parts and intersection profile part.

For planar plates that lie on a major grid plane, names the view using *<plate type> <grid plane name>*"+ or -"*<offset from grid plane>*. For example, for a transverse plate defined by a grid plane of F234, the view is named **Transverse F234**. For a grid plane of F234 and a global coordinate system offset of -0.5 m, the view is named **Transverse F234-0.5**.

For plates that do not lie on a major grid plane (such as inclined plates, knuckled plates, and planar orthogonal plates that are between grid planes), names the view using *<plate type> <parent plate system name>*. Collected plates are sorted alphabetically, and the first parent plate system name is used.

Name by Parts for Expansion

Uses the selected plate part name and finds the transverse side location of part. This rule is used for expansions of a single part or expansions of multiple parts that are not selected by block or assembly.

- View with an individual shell part Names the view using "SHELL EXPANSION "<PORT/STBD>" "<plate part name>. For example, SHELL EXPANSION PORT <B0.1>-SH.1501-1.
- View with an individual non-shell part Names the view using "EXPANSION "<plate part name>. For example, EXPANSION <B0.1>-47_20223LS.102-1.
- View with multiple shell parts not selected by block/assembly Names the view SHELL EXPANSION - PORT or SHELL EXPANSION - STBD.

Name by Query

Sheet name is the returned query name.

Name by Style

Names the view using the view definition name. For example, if the view definition used is "Mfg PinJig - Pin Data" then the view is named **Mfg PinJig - Pin Data**. For more information about view styles, see Define View Style Command (Tools Menu) in the *Drawings and Reports Reference Data Guide*.

Name by Template

This rule is applicable only to template set drawings.

The view name is the name of the template in that view.

Name for Full Expansion

This rule is applicable only to full expansion drawings.

Names the view using "SHELL EXPANSION - "<*view definition type name*>". For example, if the view definition used is "Starboard," the view is named **SHELL EXPANSION - Starboard**.

★ IMPORTANT When the criteria for a selected naming rule are not met, the software defaults to the **Default** rule.

See Also

Components Overview (on page 89)

SECTION 8

Drawings by Rule

The **Drawings by Rule** component, available in marine mode and material handling mode, creates drawings using *rule sets*. Rule sets are view styles that specify a unique way of rendering the content of a drawing. After creating a **Drawings by Rule** component, you can add rule-based drawing view types, then save the component as a package or create drawings directly.

ViewTypes

A **ViewType** contains the instructions to define a drawing view. It specifies the filter, styles, orientations, and scales used to define the drawing view. It also helps you assign inputs needed to generate the drawing view.

A ViewType contains:

- ViewStyles Specifies a unique way of rendering the content of a drawing view.
- Filter (Query) Determines the primary elements in the drawing view.
- Orientation Lists allowable orientations for views of this type.

For information on creating rule set view styles, see *Define View Style Command (Tools Menu)* in the *Drawings and Reports Reference Data Guide*.

■ NOTE Depending on the nature of the drawing, some of the ViewType definitions allow multiple selections. For example, a Hull Line ViewType allows one query and one style, but it allows multiple orientations. Another example is the Shell Expansion by Blocks ViewType that allows multiple blocks and one view is created for each selected block.

You can perform several tasks after creating a **Drawings by Rule** component. These tasks are available on the shortcut menu when you right-click the component.

Setting Up a Drawings by Rule Component

If you are an administrator, use the **Setup** command on the component shortcut menu to define views, reports, and queries for the drawings documents. For more information, see *Setup* (*Drawings by Rule Component*) (on page 104).

Saving a Drawings by Rule Package

Use the following commands to select an existing component, edit the drawing template, and save the component as a package.

- Edit Template Opens the component template in SmartSketch Drawing Editor so you can set document and sheet properties and design the layout of the drawing output. For more information, see Edit Template (Drawings by Rule) (on page 128).
- Save Package Saves the entire component as a package so it can be used for other drawing generation tasks. For more information, see Save Package Command (on page 76).

Creating Drawings and Updating

The following commands are available on the shortcut menu for a Drawings by Rule component.

- Create Drawing Creates the drawing document(s). For more information, see Create Drawing(s) Command (on page 39).
- **Refresh** Refreshes the content of the associated document views. For more information, see *Refresh* (*Shortcut Menu*) (on page 79).
- Update Updates the drawing document content. For more information, see Update (on page 83).

■ NOTE If you are using batch processing for your documents, the **Batch** command will be on the shortcut menu for the selected component or document. For more information, see Batch Processing.

Comparing 2D Drawing Objects to 3D Model Objects

You can open a Drawings by Rule document in a 3D task and compare the 2D drawing objects in the drawing document to 3D model objects. For more information, see *Compare 2D Drawing Object to 3D Model Object* (on page 362).

Other Tasks

- **Print** Prints the selected drawing. For more information, see *Print Command* (on page 45).
- Properties Displays the Properties dialog for the selected folder, component, or drawing view. For more information, see Properties Command (on page 48).
- Delete Deletes the selected folder, component, or drawing view. For more information, see Delete Command (on page 39).
- **Rename** Allows you to rename the selected folder, component, or drawing view. For more information, see *Rename Command* (on page 69).
- Copy Allows you to copy the selected item from one folder to another. For more information, see Copy Command (on page 38).
- Paste Allows you to paste the selected item from one folder to another. For more information, see *Paste Command* (on page 43).
- Modify Border Templates for Individual Sheets The Edit Sheet Properties command allows you to change the border for an individual sheet to a template different from the border template defined in the component. For more information, see Change the Border for an Individual Drawings by Rule Sheet (on page 397).

See Also

Assembly Drawings (on page 129)
Hull Lines Drawings (on page 137)
Manufacturing Drawings (on page 144)
Scantling Drawings (on page 153)
Shell Expansion Drawings (on page 173)

Setup (Drawings by Rule Component)

Defines component views, reports, and queries for creating drawings by rule. This command is available on the shortcut menu for a Drawings by Rule component on the **Management Console**.

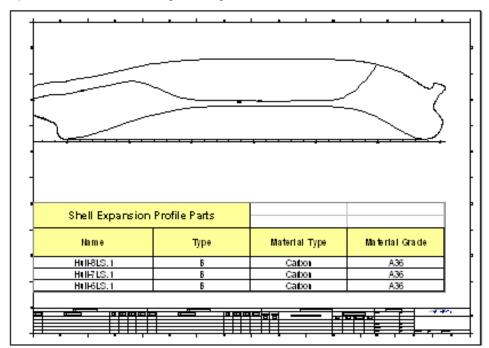
★ IMPORTANT You do not use most of the options in this command in the typical workflow for creating drawings and reports. An administrator typically performs setup. The exception is when you want to limit the number of view to be drawn. For more information, see *View Query Results* in *Queries Tab (Setup Dialog Box - Drawings by Rule)* (on page 115).

You can add as many views and reports as you like to a Drawings by Rule component. Reports are associated to views. Queries are used to populate the views for the drawing documents.

For more information, see Setup Dialog Box (Drawings by Rule Component) (on page 108).

Dependent Reports

You associate reports to drawings by rule views, such as the Shell Expansion Profile Parts report added to the following drawing.



Set up a Drawings by Rule package

The following procedure steps you through creating a new Drawings by Rule component, setting up its definition, and saving it as a package for multi-purpose usage. The procedure below creates an example Hull Lines drawing component.

Before you create the Drawings by Rule component, you might want to create a custom rule set view style or modify an existing style to use with the Drawings by Rule component. For more information on creating rule set view styles, see *Define View Style Command (Tools Menu)* in the *Drawings and Reports Reference Data Guide*.

- 1. Right-click any drawing folder under the main model node and select **New**.
- 2. On the **Ship** tab of the **Add Component** dialog box, select **Drawings by Rule**. Click **OK**. The new **Drawings by Rule** component displays under the drawing folder. To change the name of the new component, right-click the component, and select **Rename**.

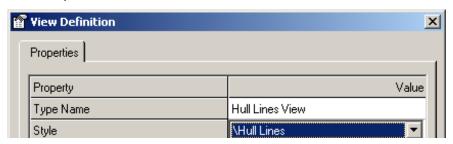
For more information, see Add a Drawings by Rule component (on page 42).

 Right-click the Drawing by Rule component and select Setup to define the views, reports, and queries used by the component. The Setup dialog box displays with the General tab displayed so you can add views and reports to the View Builder. For more information, see General Tab (Setup Dialog Box - Drawings by Rule) (on page 108).



4. On the **General** tab, click **Add View Type** 4 to add a new view to the component. The **View Definition** dialog box displays so you can define the view name, style, and other properties. For more information, see *View Definition Dialog Box* (on page 109). The views are added to the **View Builder** hierarchy.

For example, add a Hull Lines view, and define it to use the Hull Lines rule set view style.



The Hull Lines View is added to the View Builder.



5. After you create the views for the component, you can add reports to the views. Select a view, and click **Add Report** . Select a report from those shown in the **Select Template** dialog box, and click **OK** to add the report to the view.

6. You can also add some reports as embedded standalone reports that are not associated with a view. Select **View Builder**, and click **Add Report**. Select a standalone report from those shown in the **Select Template** dialog box, and click **OK** to add the report.

The report is added to the **View Builder**.

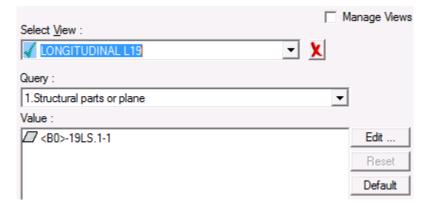


For more information, see Embedded Standalone Reports in the Report User's Guide.

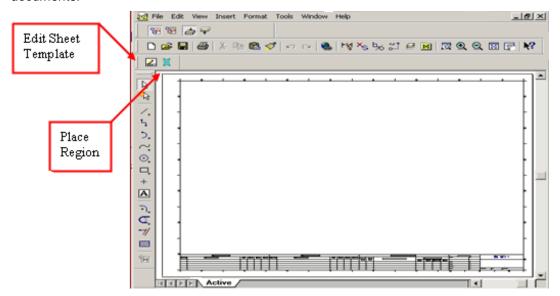
- 7. To change the properties for a report or a view, select the item in the **View Builder** hierarchy, and click **Properties**
- 8. When you are done defining views and reports, go to the **Queries** tab to define the queries for the component. For more information, see *Queries Tab (Setup Dialog Box Drawings by Rule)* (on page 115). On the **Queries** tab, select a view type from the **Select View Type** drop-down. The view types are defined on the **General** tab of the **Setup** dialog box. Select **Default** in this drop-down to specify a default filter definition to use for all view types, or you can select individual view types and define filters for each.
- 9. Change or modify the filter used for the selected view type. For example, select the Hull Lines View, then select Create New Filter in the Filter drop-down to create a new filter definition, or you can select More in the drop-down to select from a list of existing filters. Click Properties to modify the properties of the currently selected filter. The example below shows that a Coordinate System Filter is used.



- 10. Set the collation rule using the **Select Collation Rule** drop-down. The default rule is **CollateToMany**, which means that the drawings will have one part per view.
- 11. Click **Apply Filter** to run the filter definition. The rest of the dialog updates with additional definition information.



- 12. You can further define the view definition by changing the query or values associated with each view generated by the filter. For more information, see *Queries Tab (Setup Dialog Box Drawings by Rule)* (on page 115).
- 13. After you have defined all of the views, reports, and queries for the Drawings by Rule component on the **Setup** dialog box, click **OK** to save the changes.
- 14. Specify the sheet and document templates to use when creating the drawings. Right- click the Drawings by Rule component and select Edit Template. SmartSketch Drawing Editor opens with the Edit Sheet Properties dialog box displayed (Edit Sheet command). Use this command to define the properties for the sheet and the document associated to the Drawings by Rule component. For more information, see Edit Template (Drawings by Rule) (on page 128).
- 15. Use the **Edit Sheet Properties** and **Place Region** commands to set up the layout for the drawing area in the template. For more information on these commands, see the *SmartSketch Drawing Editor Help*.
 - NOTE Regions are only needed if you want views to be placed on the sheet automatically when the drawing is updated. Regions are not required. Without regions, you can drag views onto the sheet and place them as needed for multi-sheet drawing documents.



- 16. Save any changes you make to the template, and exit SmartSketch Drawing Editor.
- 17. If you are an administrator with write permissions to the SharedContent folder, you can save the current Drawings by Rule component definition into the model as a package so that it displays in the **Add Component** dialog box and can be reused by others. Right-click the component and select **Save Package**. For more information, see *Save a package* (on page 77).
- 18. You can also create the drawings for the Drawings by Rule component by right-clicking the component and selecting Create Drawings. This creates the drawings documents and displays them in the Detail View.

If you make changes to the model that impact the drawing content, the drawings will be out-of-date and you will need to refresh or update them to update the drawing content. For more information, see *Updating Documents* (on page 78).

Setup Dialog Box (Drawings by Rule Component)

Sets views, reports, and queries for a Drawings by Rule component. This dialog box displays when you select **Setup** on the shortcut menu for a Drawings by Rule component.

General Tab (Setup Dialog Box - Drawings by Rule) (on page 108) Queries Tab (Setup Dialog Box - Drawings by Rule) (on page 115)

See Also

Setup (Drawings by Rule Component) (on page 104)

General Tab (Setup Dialog Box - Drawings by Rule)

Adds views and reports to a Drawings by Rule component. This is the default tab for the **Setup** dialog box, which is displayed when you select **Setup** on the shortcut menu for a Drawings by Rule component.

If this is a new Drawing by Rule component, the **General Tab** contains no views or reports in the **View Builder** hierarchy.



If the Drawings by Rule component already has view and report definitions, the **View Builder** hierarchy displays the current view definition:



View Builder

Shows a hierarchy of the current view and report definition for the Drawings by Rule component.

M Add View Type

Adds a view to the **View Builder** hierarchy. This option is available when you select the **View Builder** root. When you click **Add View Type**, the **View Definition** dialog box displays so you can define the view type properties. For more information, see *View Definition Dialog Box* (on page 109).

Add Report

Adds a report in the following ways:

Adds a report to the selected view in the View Builder hierarchy. When you click Add Report , the Select Template dialog box displays so you can choose a dependent report for the selected view. Select a report, and click OK to add it to the view in the View Builder.

Adds an embedded standalone report in the View Builder hierarchy. An embedded standalone report is not dependent on a view. When you click Add Report , the Select Template dialog box displays so you can choose a standalone report. Select a report, and click OK to add it to the View Builder.



Delete

Deletes the currently selected view or report.

Properties

Displays the **View Definition** dialog box so you can modify view or report definitions as needed.

See Also

Setup (Drawings by Rule Component) (on page 104) Setup Dialog Box (Drawings by Rule Component) (on page 108)

View Definition Dialog Box

Defines view properties for view in a Drawings by Rule component. This dialog box displays when you add a new view to a Drawings by Rule component or edit the properties for an existing view. For more information, see *General Tab (Setup Dialog Box - Drawings by Rule)* (on page 108).

The properties for a view definition are as follows:

Type Name

Names the view type you are adding or editing.

Style

Specifies the rule set view style to use for this view. For more information on creating rule set view styles, see *Define View Style Command (Tools Menu)* in the *Drawings and Reports Reference Data Guide*.

View Naming Rule

Specifies the naming rule used for this view type. For more information, see *Labels Tab* (*Edit Ruleset View Style Dialog Box*) in the *Drawings and Reports Reference Data Guide*.

Coordinate System

Specifies the coordinate system to use with regard to the drawing view contents. These are the coordinates used for the annotation of the view. The coordinates listed are defined for the model. For **Local Coordinate System**, the main object of the view is used.

Scale Family

Specifies a scale family for the drawing view, such as Metric or Imperial. When you select a scale family, the **Scale Value** property values update.

Scale Value

Shows the scale values available for the currently selected **Scale Family**.

Orientation Rule

Specifies the rules available for orientations for the current view type. The **Orientation Rule** drives the values for the **View Direction** and **Up Direction** properties. For more information, see *Orientation Rules* (on page 110).

View Direction

Defines the "looking" direction of the view. The values change based on the selected coordinate system. As an example, for planar views with a local coordinate system, select Along -X, Along +X, Along +Y, Along -Y, Along +Z, or Along -Z. For isometric views with a local coordinate system, select Iso - Along -Z +Y -X, Iso - Along -Z +Y +X, Iso - Along -Z -Y -X, Iso - Along -Z -Y +X, Iso - Along +Z -Y +X, Iso - Along +Z -Y +X.

Up Direction

Defines the direction that is pointing up in the view, such as along the positive Y-axis.

VHL Precision

Type a value between .001 and 1.0 only if you want to reduce the precision of VHL (vector hidden line) calculations for the view and override the software-wide value. By default, the **VHL Precision** value is not set and a very small software-wide value of 1 x 10⁻⁴ is used for VHL calculations.

Geometry Validation

Select **On** to analyze geometry and discard invalid geometry. Discarded geometry is not drawn in the view. Use with caution, because invalid geometry may represent legitimate model items that are removed from the drawing. The default is **Off**.

See Also

Setup (Drawings by Rule Component) (on page 104) Setup Dialog Box (Drawings by Rule Component) (on page 108) General Tab (Setup Dialog Box - Drawings by Rule) (on page 108)

Orientation Rules

An orientation rule specifies view orientations for the current view type. The **Orientation Rule** drives the values for the **View Direction** and **Up Direction** properties. The following orientation rules are delivered:

Global Coordinate System (ISGlobalLCS.Default)

Orientation is relative to the global coordinate system. View direction options include planar view options such as **Looking Aft** or **Looking Port**; and isometric view options such as **Up Port Aft** and **Plan Starboard Forward**. Up direction options are dependent on the selected view direction and include planar view options **Plan**, **Up**, **Aft**, **Forward**, **Port**, or **Starboard**; and isometric view options **Up**, **Down**, **Left**, or **Right**.

Local Coordinate System

Orientation is relative to the local coordinate system of the part as defined by the software. View direction options include planar view options such as **Right (Along -X)** or **Top (Along +Z)**, and isometric view options such as **Top Front Right** and **Bottom Back Left**. Up

direction options are dependent on the selected view direction and include planar view options along the X-, Y-, and Z-axes in both directions, and isometric view options **Bottom**, **Top**, **Back**, or **Front**.

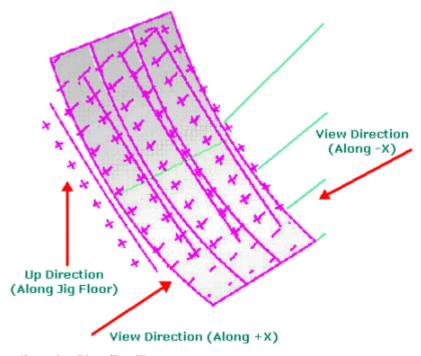
Not Applicable (SMNotApplicableLCS.Default)

Orientation rule has no significance. Both View direction and Up direction options are **Not Applicable**.

Mfg PinJig SideView (SMMfgPinJigSideLCS.Default)

Orientation is relative to PinJig local coordinate system. View direction options include planar view options such as **Left (Along +X)** or **Right (Along -X)**. The up direction option includes **Jig Floor Up**.

The up direction is along the jig floor and view directions are along **+X** and **-X**, as shown below:



Scantlings for Plate/Profile

Orientation is relative to a view direction that is across the local x-, y-, and z - axes of the view coordinate system. If the view does not have any coordinate system, the view direction will be set to the global X-, Y- or Z-axis. The following are standard combinations of view direction and up direction:

Transverse Bulkheads

Section view Looking Aft: View direction is looking aft (**Along -Z**), Up direction is up (**Along +Y**).

Section View Looking Fwd: View direction is looking forward (**Along +Z**), Up direction is up (**Along +Y**).

Longitudinal Bulkheads

Elevation Looking Stbd: View direction is looking starboard (**Along -Z**), Up direction is up (**Along +Y**).

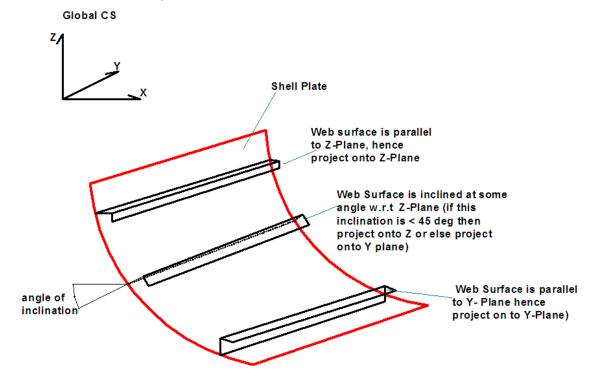
Elevation Looking Port: View direction is looking port (**Along +Z**), Up direction is up (**Along +Y**).

Decks

Plan Looking Down: View direction is looking down (**Along -Z**), Up direction is port (**Along +Y**). The forward end of the ship is always oriented to the right of the drawing.

Shell Profiles

View direction will be either **Global –Z** or **Global +Y** based on the angle between the web surface of the shell profile and **Global –Z** axis as shown below:



The orientation rule sets the view direction to:

- Global -Z (Plan looking Down) if the angle of inclination is less than the tolerance angle.
- Global +Y (Looking Port) if the angle of inclination is more than the tolerance angle.

The default value for the tolerance angle is set to **75 degrees**. You can configure the tolerance angle value using the **Angle for Target Evaluation (Optional)** view query in the view style. For more information about view queries, see Steel Order Drawing Rule Set.

Mfg Template At Index

Orientation is relative to the local coordinate system of a template within a template set. The template is identified by its index number within the template set. View direction options include planar view options such as **Right (Along -X)** or **Top (Along +Z)**, and isometric view options such as **Top Front Right** and **Bottom Back Left**. Up direction options are dependent on the selected view direction and include planar view options along the X-, Y-, and Z-axes in both directions, and isometric view options **Bottom**, **Top**, **Back**, or **Front**.

Shell Expansion for Plate

Used for partial shell plate expansions and expansion of curved and knuckled internal plates. Orientation is relative to the global coordinate system. Up direction option is **Up**. View direction options are as follows:

- Looking Outboard and Looking Inboard
- Looking Port and Looking Starboard

Expansion (Hull)

Used for full shell expansions. Orientation is relative to the global coordinate system. In all view directions, hidden line conventions are ignored. The hull plate is always solid, and all connected plates and profiles are dashed. The up direction option is **Up**. View direction options are as follows:

- Centerline Looking Port and Starboard Looking Centerline Forward is to the right in the view.
- Centerline Looking Starboard and Port Looking Centerline Forward is to the left in the view.

View Coordinate System

Orientation is relative to the coordinate system specified in the view definition. View direction options include planar view options, such as **Right** (**Along -X**) or **Top** (**Along +Z**), and isometric view options, such as **Top Front Right** and **Bottom Back Left**. Up direction options are dependent on the selected view direction and include planar view options along the X-, Y-, and Z-axes in both directions, and isometric view options **Bottom**, **Top**, **Back**, or **Front**.

If the View direction does not have any coordinate system, then it will be set to the global X-, Y- or Z-axis.

Scantling By Plate Normal

Orientation is relative to plate normal. The following are standard combinations of view direction and up direction:

- Plate normal inclined towards global X-axis
 - Looking Aft: View direction is looking aft along the plate normal (Along –Z), Up direction is up (Along +Y).
 - Looking Forward: View direction is looking forward (Along +Z), Up direction is up (Along +Y).
- Plate normal inclined towards global Y-axis
 - Looking Stbd: View direction is looking starboard along the plate normal (**Along –Z**), Up direction is up (**Along +Y**).
 - Looking Port: View direction is looking port (Along +Z), Up direction is up (Along +Y).
- Plate normal inclined towards global Z-axis
 - Plan Looking Down: View direction is looking down (Along –Z), Up direction is port (Along +Y).

The following table shows how orientation rules are typically used depending on the type of drawing you are creating:

Type of Drawing	Orientation Rules
Scantling Drawings, including the following delivered packages: Steel Order (Expansion) Steel Order by Block or Assembly Steel Order (Shell Profiles)	Global Coordinate System Scantlings for Plate/Profile (for planar plates and Shell Profiles) Shell Expansion for Plate (for shell plates)
Shell Expansion	Expansion (Hull)
Hull Lines Drawings, including the following delivered packages: Hull Lines	Global Coordinate System
Assembly Drawings, including the following delivered packages: Assembly Method Assembly Sequence	Local Coordinate System
Manufacturing Template Drawings, including the following delivered packages: Mfg Template Set	Local Coordinate System Mfg Template at Index
Manufacturing PinJig Drawings, including the following delivered packages: Mfg PinJig	Local Coordinate System
Manufacturing Profile Sketch Drawings, including the following delivered packages: Mfg Profile Sketch Mfg Profile Sketch (Multiple)	Not used
Manufacturing Templateset Full Scale Drawing	Not Applicable
Manufacturing PinJig Side View Drawing	Mfg PinJig SideView

See Also

View Definition Dialog Box (on page 109) General Tab (Setup Dialog Box - Drawings by Rule) (on page 108)

Queries Tab (Setup Dialog Box - Drawings by Rule)

Adds queries to a Drawings by Rule component. Before adding queries, you should add views and reports on the **General** tab. For more information, see *General Tab (Setup Dialog Box - Drawings by Rule)* (on page 108).

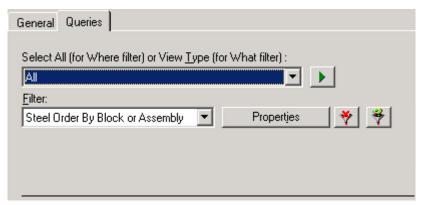
View Type Filter and Query Definition

The view type filter definitions drive the views created when the query runs.

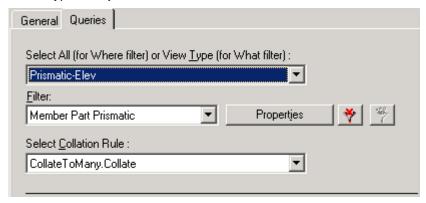
Select All (for Where filter) or View Type (for What filter)

Shows a list of the view types available for query definition. Select one of the following options:

All - Specifies objects to collate based on their location, not their type. You must select
a where filter for the location in the model used for all view definitions in the component.



An individual view type - Specifies the object types to collate. The available view types
are defined on the **General** tab of the **Setup** dialog box. Optionally, select a *what* filter
for the type of object. You must also select a collation rule.



Run Query

Runs the current query definition with the specified filter to populate the rest of the query parameters. The **Queries** tab updates with all the necessary query definition fields.

Filter

Specifies the filter to use for this query definition. Select More in the dropdown to display the

Select Filter dialog, and choose from a list of available filters. Select **Create New Filter** to create a new filter with the **Filter Properties** dialog box. The dropdown also provides a list of previously selected filters. For more information, see *Select Filter Dialog Box* (on page 122).

■ NOTE Some queries are run frequently but need customized values for the filter, called an asking filter. Select **User of filter supplies value** on the **Filter Properties** dialog box to create an asking filter. For example, select **BO**, **Include nested objects**, and **User of filter supplies value** on the **Filter Properties** dialog box. The filter will allow selection of a block or assembly when **Ask Filter Inputs** ** is clicked.

Properties

Displays the **Filter Properties** dialog box so you can modify the currently selected filter as needed. For more information about filters, see *Filter Properties Dialog Box* (on page 164).

脊 Remove Filter

Removes the specified filter from the currently selected view type query definition.

🕈 Ask Filter Inputs

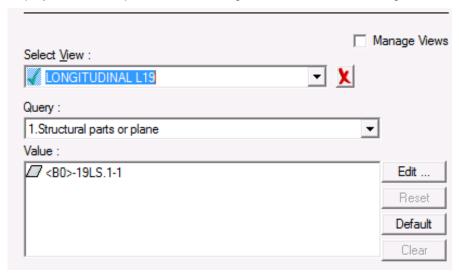
Displays the **Filter Properties for Asking Filter** dialog box, which allows selection of a value for the asking filter. The dialog is the same as the **Filter Properties** dialog box, except that only the appropriate tabs for the asking filter are displayed. This command is available only when the filter has an asking filter specified. For more information about filters, see *Filter Properties Dialog Box* (on page 164).

Select Collation Rule

Specifies how model objects are gathered into views when the drawings for this Drawing by Rule component are created. Collation rules are part of the reference data in the Catalog. For more information, see *Collation Rules* (on page 123).

View Query Results

When you click **Run Query** against the current view type query definition, the view queries display in the lower portion of the dialog box. You can make changes to the results as needed.

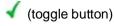


Manage Views

Allows you to select multiple views in **Select View** that can be created (using \checkmark) or deleted (using \checkmark).

Select View

Shows the views created when the view type query ran. If this drop-down is empty, you need to click **Apply Filter** to run the query and populate the view query definitions.



Displays that the views will be created in Create Drawings.

(toggle button)

Displays that the views will not be created in **Create Drawings**.

Query

Specifies the value for a query applied to the selected view. Each view can have more than one query applied. You can select each query and update the values.

A view query specifies the type of model object or parameter value used by the view definition. The list of available queries is specific to the selected package. The drawings by rule queries are most commonly used.

Drawings by Rule Queries

1. Any system, part, or reference plane

Defines the query by the selected outfitting or structural systems or parts, or a selected reference plane. When systems are selected, the parts belonging to the systems are used. Select objects that support the type of drawing view you want to create. A value for this query is required. The following can be set:

- Plate parts or systems.
- Profile parts or systems.
- Member parts or systems.
- Outfitting parts or systems, including pipe lines, pipe runs, pipe parts, hangers and supports, and equipment.
- Reference planes from any coordinate system.

Selections can be made manually using **Associate Objects to Views** in SmartSketch Drawing Editor. Selections can also be specified automatically as output of an applicable collation rule. For more information, see *Collation Rules in Drawings by Rule* in the *Drawings and Reports Reference Data Guide* or *Collation Rules* (on page 123) in the *Orthographic Drawings User's Guide*.

2. Boundary (Block, volume, or assembly)

Defines the boundaries of a query by a selected planning block, volume, assembly, or assembly block. A value for this query is optional.

3. Gathering rule

Defines the type of parts gathered in the drawing view. The rule provides a definition of the objects to be gathered and drawn in a view. The gathering rule starts with the inputs

selected in **Any system**, **part**, **or reference plane** and adds more objects to the view based on their relationships to each other. This option opens the **Select Gathering Rule** dialog box. Select a rule that supports the type of drawing view you want to create, such as **Steel Order** or **Assembly**. A value for this query is required. The default value is **Steel Order**.

4. Gathering method

Defines how model objects are included in the drawing view. The volume selected with **Block**, **volume**, **or Assembly** or created with **Volume creation rule** is used as input for the gathering method.

- Gather related objects in volume Parts are gathered based on connections to the selected parts resulting from Any system, part, or reference plane. If a volume is defined for Volume creation rule, only connected objects in the volume are gathered.
- Gather all objects in volume All parts in the volume are gathered even if they are not connected to the selected parts resulting from Any system, part, or reference plane. If a volume is not defined for Volume creation rule, this option is not valid. This is the default value.

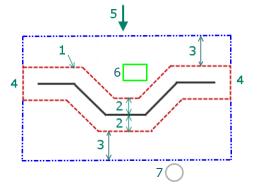
Relationships (such as parent/child or bounded/bounding) for all gathered objects are also gathered and used by the selected **Gathering rule**.

5. Volume creation rule

Defines the volume for the view by one of the following methods:

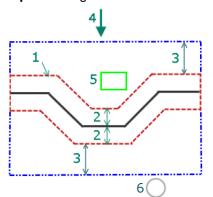
Create volume along surface of input parts with boundary extents - The volume follows the surfaces of the selected parts resulting from Any system, part, or reference plane. The volume depth (normal to the view) is extended by the values of Volume growth into drawing and Volume growth out of drawing. The volume area (in the plane of the view) is extended to (or clipped by) the boundaries defined by Boundary (Block, volume, or assembly). The gathering range is extended by the values defined by Extend Into and Extend Out Of.

Example: Corrugated bulkhead



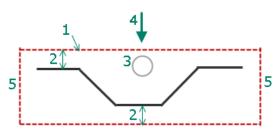
- 1 Cross-section of volume
- 2 Volume growth
- 3 Extended gathering range
- 4 Volume extended to boundaries
- 5 View direction
- 6 Additional part gathered into view
- 7 Additional part not gathered into view
- Create volume along surface of input parts without boundary extents The volume follows the surfaces of the selected parts resulting from Any system, part, or reference plane. The volume depth (normal to the view) is extended by the values of Volume growth into drawing and Volume growth out of drawing. The volume area (in the plane of the view) is not extended to (nor clipped by) the boundaries defined by Boundary (Block, volume, or assembly). The gathering range is extended by the values defined by Extend Into and Extend Out Of.

Example: Corrugated bulkhead



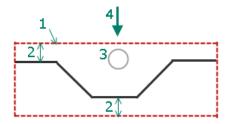
- 1 Cross-section of volume
- 2 Volume growth
- 3 Extended gathering range
- 4 View direction
- 5 Additional part gathered into view
- 6 Additional part not gathered into view
- Create volume from range box around input parts with boundary extents The volume is a rectangular range box around the selected parts resulting from Any system, part, or reference plane. The volume depth (normal to the view) is extended by the values of Volume growth into drawing and Volume growth out of drawing. The volume is extended to (or clipped by) the boundaries defined by Boundary (Block, volume, or assembly). See the corrugated bulkhead example below.

Example: Corrugated bulkhead

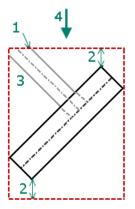


- 1 Cross-section of volume
- 2 Volume growth
- 3 Additional part gathered into view
- 5 4 View direction
 - **5** Volume extended to boundaries
- Create volume from range box around input parts without boundary extents The volume is a rectangular range box around the selected parts resulting from Any system, part, or reference plane. The volume depth (normal to the view) is extended by the values of Volume growth into drawing and Volume growth out of drawing. The volume is not extended to (nor clipped by) the boundaries defined by Boundary (Block, volume, or assembly). See the corrugated bulkhead example below.

Example: Corrugated bulkhead



Example: Non-orthogonal member



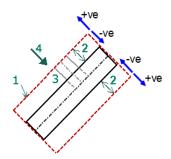
- 1 Cross-section of volume
- 2 Volume growth
- 3 Additional part gathered into view
- 4 View direction

Create oriented volume around input parts without boundary extents - The volume is a rectangular box around the parts. The box is oriented around parts to minimize the volume. The parts are typically not orthogonal, but the view direction is typically normal to the box. The volume depth (normal to the view) is extended by the values of Volume growth into drawing and Volume growth out of drawing.

■ NOTES

- If the view direction is not normal to the box, then the volume is extended to (or clipped by) the boundaries defined by Boundary (Block, volume, or assembly) instead of Volume growth into drawing and Volume growth out of drawing.
- If the view direction is normal to the box, then the volume is extended to (or clipped by) **Volume growth into drawing** and **Volume growth out of drawing**. These values can be positive or negative.

Example: Non-orthogonal member



- 1 Cross-section of volume
- 2 Volume growth
- 3 Additional part gathered into view
- 4 View direction

Do not create volume - A volume is not used to restrict gathering using gathering rule.
 For example, a shell longitudinal profile view does not use a volume. It instead uses levels to gather secondary parts.

NOTE The volume selected with **Boundary (Block, volume, or assembly)** or created with **Volume creation rule** is also used for view clipping in the graphic rule. For more information, see *Graphic Rules in Drawings by Rule* in the *Drawings and Reports Reference Data Guide*.

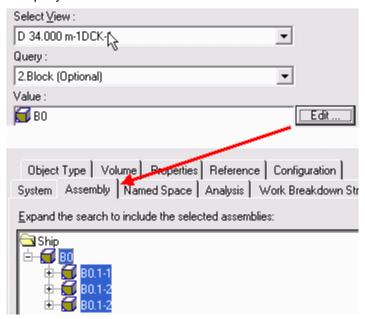
For other queries, see the appropriate drawing type under *Drawings by Rule View Style Rule Sets* in the *Drawings and Reports Reference Data Guide*.

Value

Identifies the values generated when you ran the query using the **Apply Filter** button.

Edit

Allows you to edit the selected query value. In the following example, **Query** is changed to **Block (Optional)**. Click **Edit** to select a new filter (from the **Assembly** tab) as the value for the query.



Reset

Resets the values back to their original state for the specified query.

Default

Sets the current value as the default for the current view query definition.

Clear

Removes assemblies or blocks in the **Value** list. You can then click **Filter** again, and select new values for the query. This option is available only when assemblies or blocks are selected in **Query**.

See Also

Setup (Drawings by Rule Component) (on page 104) Setup Dialog Box (Drawings by Rule Component) (on page 108)

Select Filter Dialog Box

Creates, edits, deletes, and selects filters for use with the **Define Workspace**, **Surface Style Rules**, and other **Select by Filter** commands, including Project Management's **Model Data Reuse** (MDR), Drawings View Styles, and Reports commands that require runtime filter selection. You can access this dialog box in several ways.

- Select File > Define Workspace, and select the More option in the Filter box.
- Select Format > Surface Style Rules, click New or Modify, and then select the More option in the Filter box.
- Select Tools > Select by Filter.

The tree view displays the following types of filters:

- Catalog Filters These filters are used to reference data in the Catalog. For example, a
 catalog filter could apply to company-wide operations. Your administrator can define
 Company_Filter_1, Company_Filter_2, and so forth.
- Model Filters These filters are available to everyone assigned to a specific model database. There are delivered catalog filters to query on the different types of model objects. You must have the appropriate privileges to create, edit, or delete these filters.
- My Filters These are personal filters that you create and place in the My Filters folder. They are visible only to you, the owner. You cannot see the personal filters of others, and they cannot see your personal filters. Select a filter from one of the listed filters, or create a new filter to meet your specific requirements.

New Folder

Creates a new folder.

New Filter (Simple or Asking)

Displays the **New Filter Properties** dialog box so that you can create a new filter. Asking filters allow you to specify the parameters of the search. An asking filter has built-in functionality to ask for values (with boxes that you are required to supply). The values apply to properties that you have already designated you will supply when the filter runs. Asking filters are portable between models.

▶ NOTE Model Data Reuse (MDR) does not support asking filters. The only valid filter types for an MDR transaction are System, Permission Group, Object Type, Volume and Properties. You can define the filter on any one of these tabs or in a combination using multiple tabs.

New Compound Filter

Displays the **New Compound Filter Properties** dialog box, which you use to create a new compound filter containing the Or, And, or Not operators. Compound filters are not supported for MDR.

New SQL Filter

Displays the **New SQL Filter Properties** dialog box, in which you can type the text of an SQL query. SQL filters are not supported for MDR

× Delete

Removes a filter or folder from the **Select Filter** list. If you delete a folder, the software also deletes its contents.

Rename

Changes the name of an existing filter or folder from the Select Filter list.

Properties

Displays the **Filter Properties** dialog box so that you can select the properties that determine your filter search criteria.

■ NOTES

- If this dialog box is activated from the Select by Filter command, you can select multiple filters on this dialog box. Hold CTRL or SHIFT, and click each filter. When you click OK, all objects that fit the selected filters are selected.
- If this dialog box is activated from the Select by Filter command, it clears the select set before adding objects to the select set.

Manage View from Setup Dialog Box

Create or Delete a View

- 1. Select a view from the **Select Views** drop-down menu.
- Click

 ✓ or

 X to create or delete a view respectively.
- 3. Click OK.

Create or Delete Multiple Views

1. Select the Manage Views check box.

Select Views to create is displayed.

- 2. Select or uncheck the views that you want to create or delete respectively.
- 3. Click OK.

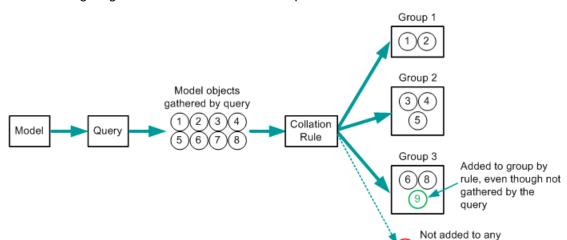
■ NOTES

- Setup also displays the views that were created from view explorer and RAD documents.
- Image displayed for each view is its current status.
 - J Displays view will be created.
 - X Displays view will not be created.
- All the changes to the views are applied the next time you create drawings.

Collation Rules

A collation rule specifies how model objects returned by a query are grouped. View styles then use the groups to create views in drawings by rule. Collation rules are part of the reference data in the catalog.

group - Does not meet criteria of rule



The following diagram shows the collation concept:

The following collation rules are delivered:

CollateAssemblySequence

Identifies the parts in an assembly and the assembly sequence of the parts as defined in the Planning task, then:

- 1. Places each part of the assembly in a separate group.
- 2. Creates a group for each step of the sequence by adding the next part(s) in the sequence.

NOTE Only groups parts within an assembly, not assemblies within an assembly or block.

CollateAssemblyStep

Identifies the parts and assemblies that make up an assembly or block and the assembly sequence of each, then creates a group for each step of the sequence by adding the next part(s) or assembly in the sequence.

CollateConnectedPlates

Groups connected plate parts. Connected plates are connected face-to-face (lapped connections) or edge-to-edge (butt connections).

CollateConnectedProfiles

Groups connected profiles parts. Connected profiles are connected end-to-end.

CollateContiguousBlocks

Groups spatially connected blocks.

CollateContiguousProfiles

Groups spatially connected profile parts.

CollateCoplanarPlates

Groups coplanar plate parts or plate parts with coplanar parent systems.

CollateForSteelOrder

Creates groups of major parts in a selected block/assembly. A grouping is created for parts that meet each of the following tests:

Major Plate Part Tests

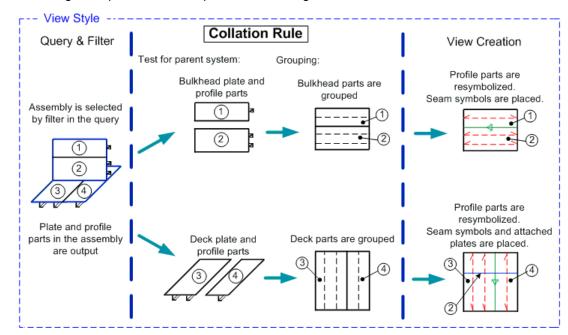
- Plate parts of the same type, such as deck plates, transverse bulkhead plates, longitudinal bulkhead plates, and hull plates.
- Plate parts belonging to the same system.
- Coplanar plate parts that may be in different systems. Coplanar means that plates with different thicknesses and thickness directions are collated if the thicknesses overlap.
- Connected plate parts or plate parts with connected parent systems. Connections must be edge-to-edge (butt connections) or face-to-face (lapped connections).

Major Profile Part Tests

- Hull profile parts.
- Profile parts belonging to the same system.
- Connected profile parts or profile parts with connected parent systems. Connections must be end-to-end.

★ IMPORTANT

- Hull plate parts are expanded and are separated into port and starboard groups.
- Brackets and standalone plate parts are not grouped because brackets (created in both the Molded Forms and Structural Detailing tasks) and standalone plates (created in Structural Detailing task) are not major parts. After drawings are created, you can manually create views for these parts.



The following example shows a simple collation using the CollateForSteelOrder rule:

CollateMemberParts

Groups standard member parts and plate parts that make up a built-up member part. A view is created for each member part.

CollateSimilarProfiles

Groups similar profiles without sketch features to a single view by matching the following conditions between profiles:

- Cross sections The cross-section type and size must be the same.
- Material grade The material grade must be the same.
- End cut counts.
- End cut smart items The end cut types must be the same, but the end cut parameters may be different.
- Corner feature counts.
- Corner feature smart items The corner types must be the same but the feature parameters may be different.
- Remaining feature counts.
- Remaining feature distances.
- Remaining feature parameter values Edge features must be exactly the same, including the position, type, and parameters.

CollateSimilarProfilesByXML

Uses information in the manufacturing profile XML created in the Structural Manufacturing task. Groups similar profiles without sketch features to a single view by matching conditions between profiles. The goal is to group profiles based on the profile XML similar to the **CollateSimilarProfiles** collation rule. However, the profile XML only contains the data to support the following conditions:

- Material grade
- End cut counts
- Corner feature counts
- Remaining feature counts
- Remaining feature distances

CollateTemplateSetToTemplates

Places each template in a template set in a separate group.

CollateToFlipShellProfilesByDir

Groups shell profiles with aft orientation into a view and reverses the view direction.

CollateToMany

Places each part in a separate group.

CollateToOne

Groups all parts.

The following table shows how collation rules are typically used, depending on the type of drawing you are creating:

Type of Drawing	Collation Rules
Scantling Drawings, including the following delivered packages: Steel Order - Expansion Steel Order by Block or Assembly	CollateToOne CollateToMany CollateConnectedProfile CollateConnectedPlates CollateContiguousProfiles CollateCoplanarPlates CollateForSteelOrder
Manufacturing Profile Sketch Drawings, including the following delivered packages: Mfg Profile Sketch Mfg Profile Sketch - Multiple	CollateToOne CollateToMany CollateSimilarProfiles CollateSimilarProfilesByXML CollateToFlipShellProfilesByDir

Type of Drawing	Collation Rules
Manufacturing Template Drawings, including the following delivered packages:	CollateTemplateSetToTemplates
Mfg Template Mfg Template Set	
Manufacturing PinJig Drawings, including the following delivered packages:	CollateToOne CollateToMany
Mfg PinJig	
Hull Lines Drawings, including the following delivered packages:	CollateToOne CollateToMany
Hull Lines	
Assembly Drawings, including the following delivered packages:	CollateAssemblySequence CollateAssemblySteps
Assembly Method Assembly Sequence	

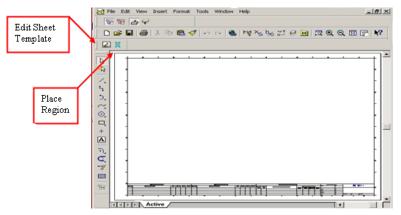
See Also

Queries Tab (Setup Dialog Box - Drawings by Rule) (on page 115)

Edit Template (Drawings by Rule)

Opens a Drawings by Rule template in SmartSketch Drawing Editor. If this is a new Drawings by Rule component with no associated template, the 2D Drawings Editor opens as a new document. You can use SmartSketch Drawing Editor commands to specify the template to use with the document or per sheet, then define regions and other definitions for the template. If the component is already associated to a template, the template opens in SmartSketch Drawing Editor so you can make changes. To use this command, select a Drawings by Rule component or package and select **Edit Template** on the shortcut menu.

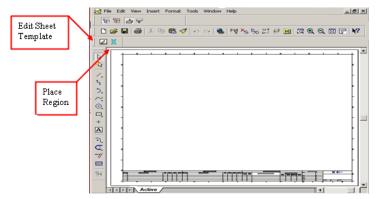
When SmartSketch Drawing Editor opens, you can edit sheet and document properties and place regions. For more information, see the SmartSketch Drawing Editor Help.



Edit a Drawings by Rule Template

This procedures show you how to edit a Drawings by Rule template. The procedure assumes you have already created a Drawings by Rule component in the Drawings and Reports task **Management Console** or in a 3D modeling task using the **Tools > Drawing Console** command.

- 1. Right-click a Drawings by Rule component and select **Edit Template** on the shortcut menu. The SmartSketch Drawing Editor application opens with the current template displayed.
- 2. Use the **Edit Sheet Properties** and **Place Region** commands to set up the layout for the drawing area in the template. For more information on these commands, see the SmartSketch Drawing Editor Help.



3. Save any changes you make to the template and exit SmartSketch Drawing Editor.

Assembly Drawings

Assembly drawings depict a collection of structure and outfitting parts assembled into a larger unit.

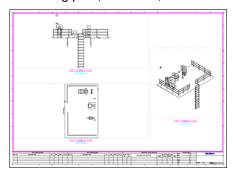
The following things must be considered when creating an assembly drawing:

 Definition of space for the drawing - You must define the space for the drawing in planning or compartmentation to generate an assembly drawing.

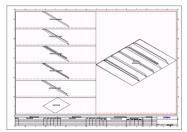
- Boundary marking If assembly boundary information exists, it must be marked on the installation drawing.
- **Shop orientation** The unit is assembled prior to installation in the ship. Therefore the drawings must have a shop orientation.

You can create assembly drawings in the following ways:

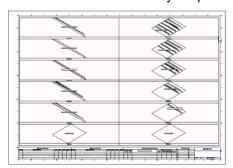
 Use the delivered drawings-by-rule Assembly component to create assembly drawings containing plan, elevation, and isometric views.



Use the delivered Assembly Method component to create assembly method drawings.



Use the delivered Assembly Sequence component to create assembly sequence drawings.



 Use the Drawings by Rule component and define it with assembly views using an assembly drawing rule set view style. For more information, see *Drawings by Rule* (on page 102).

What do you want to do?

- Create an assembly drawing (on page 131)
- Create an assembly method drawing (on page 133)
- Create an assembly sequence drawing (on page 135)

Create an assembly drawing

The following procedure creates isometric, plan, and elevation views for a pipe run, using a drawing-by-rule assembly component. The workflow is performed in the Drawings and Reports task or from **Tools** > **Drawing Console** in other tasks.

Create a drawing component

 To create a new folder, right-click the drawing root in the Management Console and select New.

The Add Component dialog box displays.

- 2. Select Folder on the General tab of the Add Component dialog box.
- 3. Click OK.
- 4. Right-click the folder, and select **Rename**. Give the folder the needed name.
- 5. Right-click the folder and select New.

The Add Component dialog box displays.

- 6. Select the Ship tab of the Add Component dialog box.
- 7. Select the **Assembly** or the **Assembly (Generic)** appackage, and then click **OK**.

The **Assembly** component displays in the Management Console. This component contains the views to be created: **Looking +X**, **Looking -Z**, and **Isometric View**.

8. Right-click the component in the Management Console, and select **Rename**. Give the component the needed name.

Set up and run queries (optional)

- ★ IMPORTANT This procedure can be skipped if you are using the default settings for the component. Perform these steps only when you need to change the queries in the component.
- 1. Right-click the component, and select **Setup**.

The **Setup** dialog box displays. For more information, see Setup Dialog Box (Drawings by Rule Component) (on page 108). The **View Builder** hierarchy contains views for each of the view types to be created.

- 2. Select the Queries tab in the Setup dialog box.
- 3. Select **All** in the **Select View Type** box.
- 4. An existing asking filter for the component displays in the **Filter** box. Select another value if needed.
- A collation rule name displays in the Select Collation Rule box. Select another value if needed.
- 6. Click Ask Filter Inputs *.

The Filter Properties for Asking Filter dialog box displays.

- 7. Select the needed system, part, assembly object, or reference plane from the hierarchy:
 - For a scantling drawing, select a structural part, structural system, or reference plane.
 - For an assembly drawing, select an assembly, block, or assemblyblock.

- For a piping drawing, select a pipeline.
- For a pipe support drawing, select a pipeline, or individual hangers and supports.

To include child objects, select **Include nested objects.** Click **OK**.

- 8. Click Run Query ▶.
- 9. Select a view in the **Select View Type** box.

Results for the view type display at the bottom of the **Queries** tab for **Views**, **Queries**, and **Values**.

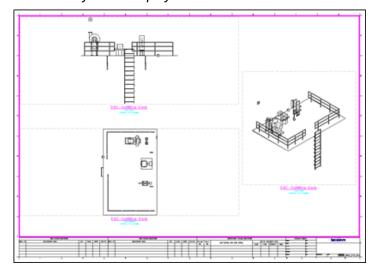
- 10. Each result for **Views** is a view based on the view styles in the component and the object types defined in **Values**.
- 11. If the parts are not the expected results, select a different filter in the **Filter** box, click **Ask Filter Inputs** to select new inputs, and again click **Run Query**.
- 12. Repeat for remaining views in the **Select View Type** box.
- 13. Click **OK**.

Create drawings

- In the Management Console, right-click the component, and select Update.
 Because you have not selected what is to be displayed on the drawings, the Filter Properties for Asking Filter dialog box displays.
- From the assembly hierarchy, select one or more blocks, assemblies, or assembly blocks.
 The Drawing Generation dialog box displays.
- 3. When the **Generation Status** field displays "Creating Documents Complete," click **Exit**.

 One drawing for each selected assembly displays in the Detail View. The drawings are up-to-date and ✓ is superimposed on each drawing icon.
- 4. Open a drawing.

All assembly views display on the same sheet.



TIPS

- If a batch server is set up by your system administrator, batch update commands are also available. For more information, see *Batch Commands* (on page 37).
- To update individual drawings, right-click each drawing, and select Update.
- When a drawing are out-of-date, X is superimposed on the drawing icon.

Create an assembly method drawing

The following procedure steps you through creating an Assembly Method drawing.

- 1. In the **Console**, right-click the root level of the hierarchy and select **New** on the shortcut menu. On the **Add Component** dialog box, select **Folder** and click **OK**.
 - TIP You could also select an existing folder in the hierarchy and skip this step.
- Right-click the folder and select New. On the Add Component dialog box, go to the Ship tab. Select an Assembly Method drawing component. For example, select the Assembly Method component and click OK. The New Assembly Method component is added to the folder. This is the component you use to create your Assembly Method drawings.
- 3. Right-click the new component and select **Setup** on the shortcut menu. The **Setup** dialog box displays.
- 4. On the **General** tab, note the **View Builder** definition. For the delivered **Assembly Method** component, one **View** is already defined.



You can add views and add views to reports as needed to define the contents of your drawings.

- 5. You can select a view or report and click **Properties** to display the **View Definition** dialog box showing the current property settings for the view or report. Change the properties as needed for the assembly method drawings you are creating. For more information, see *General Tab (Setup Dialog Box Drawings by Rule)* (on page 108).
- 6. Go to the **Queries** tab to set up the query definition for the assembly method drawings. The query definition identifies the objects in the model that will be included in the drawing. For more information, see *Queries Tab (Setup Dialog Box Drawings by Rule)* (on page 115).
- 7. In the **Select view type** dropdown, select the view type to which you want the query associated. You can define separate queries per view type or you can use a default query for multiple view types.

Specify a filter to identify the objects used in the query. For an assembly method drawing, the profiles you select do not have to be manufactured. For example, the graphic below shows that the **SeqAssy** filter is being used for the **Assembly Method** view type.



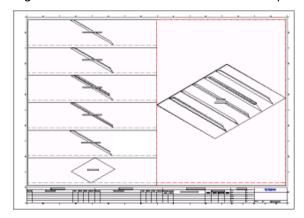
Notice that the **Collation Rule** for the **Assembly Method** setup is already set to **CollateAssemblyStep**. This collation rule identifies the parts that make up an assembly, then creates a view for each part in the assembly and one view for the "final" assembly.

8. Click **Apply Filter** to run the filter definition. The rest of the dialog updates with additional definition information.



■ NOTE You can further define the view definition using the buttons at the bottom of the **Setup** dialog box. For more information, see *Queries Tab* (Setup Dialog Box - Drawings by Rule) (on page 115).

- 9. Click **OK** on the **Setup** dialog box to save the view and query definitions.
- 10. If you need to edit the template associated with the drawing component, right click the component and select **Edit Template**. The template associated with the component opens in SmartSketch Drawing Editor so you can edit the layout and presentation for the drawings you are creating. For more information, see *Edit Template (Drawings by Rule)* (on page 128).
- 11. After you complete setup of the Assembly Method component, right-click the component and select **Create Drawings** to create the drawing documents. The new documents display in the **Detail View**. The red X icon indicates that the drawings are out-of-date (the current model information needs to be updated in the drawing views).
- 12. Right-click the component again and select **Update** to update the drawing documents with the information from the model. A dialog box shows the update progress.
- 13. When the process is complete, review the information in the dialog box for any problems that may have occurred, then click **OK** to close the dialog box. Notice that the up-to-date documents are now available.
- 14. Right-click the document and select **Edit** to open it in SmartSketch Drawing Editor.

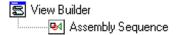


■ NOTE If you need to create a customized Assembly drawing component, create a Drawings by Rule component and define it as an Assembly drawing component. For more information, see *Drawings by Rule* (on page 102).

Create an assembly sequence drawing

The following procedure steps you through creating an Assembly Sequence drawing.

- 1. In the **Console**, right-click the root level of the hierarchy and select **New** on the shortcut menu. On the **Add Component** dialog box, select **Folder** and click **OK**.
 - TIP You could also select an existing folder in the hierarchy and skip this step.
- Right-click the folder and select New. On the Add Component dialog box, go to the Ship tab. Select an Assembly Sequence drawing component. For example, select the Assembly Sequence component and click OK. The New Assembly Sequence component is added to the folder. This is the component you use to create your Assembly Sequence drawings.
- 3. Right-click the new component and select **Setup** on the shortcut menu. The **Setup** dialog box displays.
- 4. On the **General** tab, note the **View Builder** definition. For the delivered **Assembly Sequence** component, one **View** is already defined.



You can add views and add views to reports as needed to define the contents of your drawings.

- 5. You can select a view or report and click **Properties** of to display the **View Definition** dialog box showing the current property settings for the view or report. Change the properties as needed for the assembly method drawings you are creating. For more information, see *General Tab (Setup Dialog Box Drawings by Rule)* (on page 108).
- 6. Go to the **Queries** tab to set up the query definition for the assembly sequence drawings. The query definition identifies the objects in the model that will be included in the drawing. For more information, see *Queries Tab (Setup Dialog Box Drawings by Rule)* (on page 115).
- 7. In the **Select view type** dropdown, select the view type to which you want the query associated. You can define separate queries per view type or you can use a default query for multiple view types.

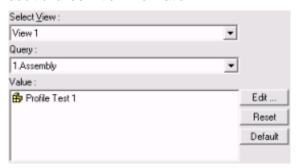
Specify a filter to identify the objects used in the query. For an assembly sequence drawing, the profiles you select do not have to be manufactured. For example, the graphic below shows that the **SeqAssy** filter is being used for the **Assembly Sequence** view type.



Notice that the **Collation Rule** for the **Assembly Sequence** setup is already set to **CollateAssemblySequence**. This collation rule identifies how the assembly is built, then

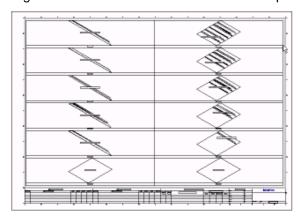
creates a view for each part of the assembly in proper sequence and a view that shows how the assembly is being built after "each" part is added.

8. Click **Apply Filter** to run the filter definition. The rest of the dialog updates with additional definition information.



■ NOTE You can further define the view definition using the buttons at the bottom of the **Setup** dialog box. For more information, see *Queries Tab* (Setup Dialog Box - Drawings by Rule) (on page 115).

- 9. Click **OK** on the **Setup** dialog box to save the view and query definitions.
- 10. If you need to edit the template associated with the drawing component, right click the component and select **Edit Template**. The template associated with the component opens in SmartSketch Drawing Editor so you can edit the layout and presentation for the drawings you are creating. For more information, see *Edit Template (Drawings by Rule)* (on page 128).
- 11. After you complete setup of the Assembly Sequence component, right-click the component and select **Create Drawings** to create the drawing documents. The new documents display in the **Detail View**. The red X icon indicates that the drawings are out-of-date (the current model information needs to be updated in the drawing views).
- 12. Right-click the component again and select **Update** to update the drawing documents with the information from the model. A dialog box shows the update progress.
- 13. When the process is complete, review the information in the dialog box for any problems that may have occurred, then click **OK** to close the dialog box. Notice that the up-to-date documents are now available.
- 14. Right-click the document and select Edit to open it in SmartSketch Drawing Editor.



■ NOTE If you need to create a customized Assembly drawing component, create a Drawings by Rule component and define it as an Assembly drawing component. For more information, see *Drawings by Rule* (on page 102).

Hull Lines Drawings

Hull Lines drawing views utilize a Hull Lines rule set view style. The Hull Lines drawings created by the component depend on the definition you set for the component. For more information, see *Drawings by Rule* (on page 102).

You can save each Hull Lines Drawing component as a package to be used or modified as needed. For more information, see Hull Lines Drawings Common Tasks.

You can create the following types of hull lines drawings:

Body Plan

Contains frame hull lines. Also includes objects that are connected to the shell, such as decks, bulkheads, profiles, and planning blocks.

You can create the following types of body plans drawings:

- From forward hull to reference plane, taken from the port side of the ship
- From reference plane to the after hull, taken from the starboard side of the ship
- Above types expressed together, with each on half of the drawing
- Complete hull body plan drawing

Plan View (waterline)

Contains waterline hull lines.

You can create the following types of plan drawings:

- From the top to a certain waterline
- From the baseline to a certain waterline
- Top and bottom combined
- Entire waterline plans

Profile View (buttock)

Contains buttock hull lines.

You can create the following types of profile drawings:

- From the centerline looking out to a certain buttock plane
- From the outside of the shell looking in to a certain buttock plane
- Looking out and looking in combined
- Entire buttock line plan

Plane Definitions

When creating hull lines drawings, you define planes intersecting the shell.

Intersection curve - The intersection curve is defined by the intersection between a reference plane and the hull.

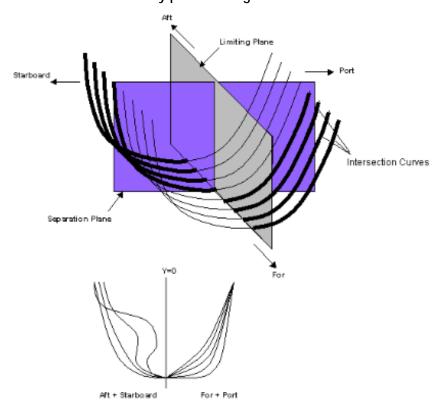
Limiting plane - This plane limits the display of the intersection curve. You do not define the limiting plane; it is used to determine how much of an intersection curve should be shown on the drawing. For body plan and plan view drawings, the system should use Y=0 as the limiting plane. For the profile view drawing, a limiting plane is not necessary because the entire intersection curve is shown on the drawing.

Separation Plane - This plane divides the hull into two sections as follows:

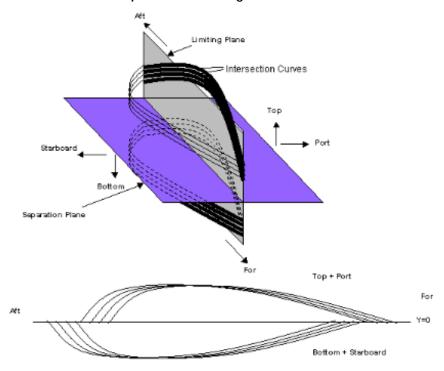
Body plan: Fore and AftPlan view: Bottom and Top

Profile view: Outboard and Inboard

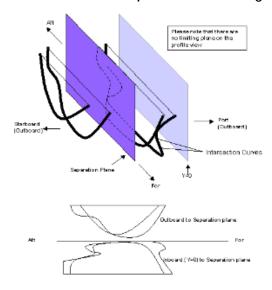
Plane definitions in body plan drawing



Plane definitions in plan view drawing



Plane definitions in profile view drawing



Line Definitions

Transverse line - Frame line on the hull created by the intersection of transverse (X-axis) grid planes with the hull.

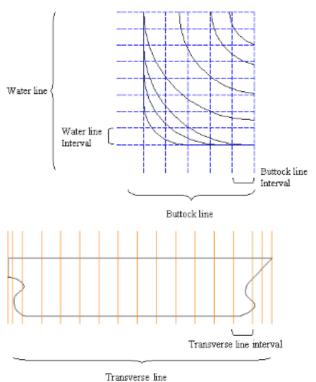
Buttock line - Longitudinal line on the hull created by the intersection of longitudinal (Y-axis) grid planes with the hull.

Waterline - Deck line on the hull created by the intersection of the deck (Z-axis) grid planes with the hull.

Line interval - The interval between intersection lines.

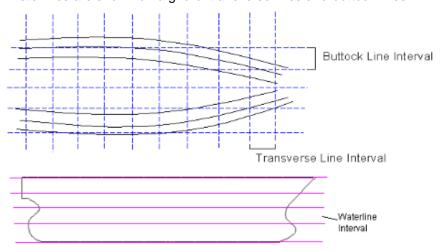
Plane definitions in body plan drawing

A body plan drawing is a transverse view of the hull. Transverse lines are shown on a grid of waterlines and buttock lines.



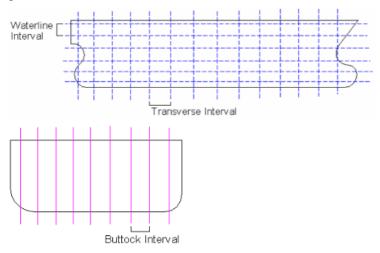
Line definitions in the plan view drawing

Waterlines are shown on a grid of transverse lines and buttock lines.



Line definitions in profile view drawing

A profile view drawing is a longitudinal elevation view of the hull. Buttock lines are shown on a grid of waterlines and transverse lines.



What do you want to do?

Create a hull lines drawing (on page 141)

Create a hull lines drawing

The following procedures steps you through creating a Hull Lines drawing using the delivered package. For more information on Hull Lines drawings, see *Hull Lines Drawings* (on page 137). For a list of common tasks for creating Hull Lines drawings, see Hull Lines Drawings Common Tasks.

1. In the **Console**, create or select a folder and add the delivered **Hull Lines** package to the hierarchy. This package displays on the **Ship** tab of the **Add Component** dialog box.

Add a Drawings by Rule component (on page 42)

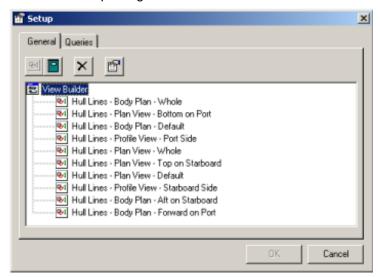


2. To edit the template defined for the package or the layout of the template, right-click the Hull Lines package and select **Edit Template** on the shortcut menu. The template associated with the package opens in SmartSketch Drawing Editor. You can edit the layout and the template association for the drawings which will be created by this package.

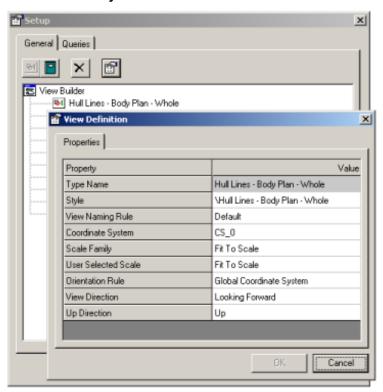
Edit a Drawings by Rule Template (on page 129)

3. Save your changes to the template and exit SmartSketch Drawing Editor.

4. Right-click the Hull Lines package and select **Setup** to display the **Setup** dialog box and the current view and query definition for the package. The **General** tab shows the views already defined for this package.

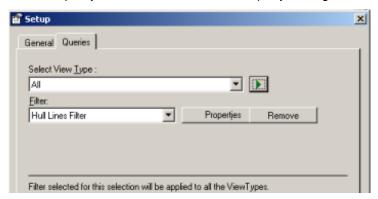


You can make changes, add reports to views, modify view and report properties, or remove views that you do not need. For more information, see *General Tab (Setup Dialog Box - Drawings by Rule)* (on page 108). For example, you could modify the view properties of the **Hull Lines - Body Plans - Whole** view:

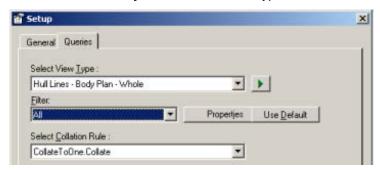


5. To change the query definition for individual view types or set a default query for all view types, go to the **Queries** tab. For more information, see *Queries Tab (Setup Dialog Box - Drawings by Rule)* (on page 115).

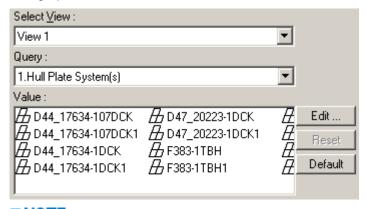
For example, you could leave the default query setting for **All** of the view types:



You could also change the query associated to each individual view as shown below with the **Hull Lines - Body Plan - Whole** view type.



6. After setting the query filter, click **Apply Filter** to run the filter definition. The rest of the dialog updates with additional definition information.



NOTE You can further define the view definition using the buttons at the bottom of the **Setup** dialog box. For more information, see *Queries Tab* (Setup Dialog Box - Drawings by Rule) (on page 115).

7. Click **OK** on the **Setup** dialog box to save your changes.

8. To save this modified package for other drawing sets, right-click the package and select **Save Package**.

Save a package (on page 77)

9. You can also create the drawings for the Drawings by Rule component by right-clicking the component and selecting **Create Drawings**. This creates the drawings documents.

If you make changes to the model that impact the drawing content, the drawings will be out of date and you will need to refresh or update them to update the drawing content. For more information, see *Updating Documents* (on page 78).

NOTE If you need to create a customized Hull Lines drawing component, create a Drawings by Rule component and define it as a Hull Lines drawing component. For more information, see *Drawings by Rule* (on page 102). You can also create customized Hull Lines rule set view style to use with your drawings. For more information, see *Define View Style Command (Tools Menu)* in the *Drawings and Reports Reference Data Guide*.

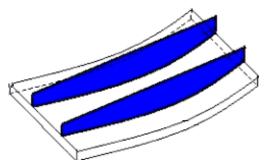
Manufacturing Drawings

Manufacturing drawings are created using the Drawings by Rule functionality. Manufacturing drawing types include the following:

- Profile Sketch A drawing that shows a two dimensional wireframe representation of a
 detailed profile that has manufacturing data applied such as margins, shrinkage, marking
 lines, and beveling. You can also create a Multiple Profile Sketch drawing, a profile sketch
 with multiple profiles per sheet.
 - **NOTE** The report for a profile sketch drawing indicates the board side of a symmetrical profile.
- Pin Jig A drawing that shows a manufacturing aid consisting of a height-adjustable pedestal that supports a part such as a plate, assembly, or block.



 Template Set - A drawing that shows a group of templates prepared for a single plate to check its curvature during and after the bending process.



You can also create customized manufacturing drawings using a Drawings by Rule component and defining it with manufacturing drawing views that utilize a Manufacturing Drawing rule set view style. For more information, see *Drawings by Rule* (on page 102).

You can save each Manufacturing Drawing component as a package to be used or modified as needed. For more information, see Manufacturing Drawings Common Tasks.

What do you want to do?

- Create a profile sketch drawing (on page 145)
- Create a multiple profile sketch drawing (on page 147)
- Create a pin jig drawing (on page 149)
- Create a template set drawing (on page 151)

Create a profile sketch drawing

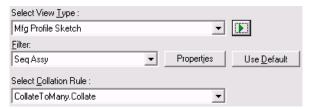
The following procedure steps you through creating a profile sketch drawing. This procedure shows how to create a profile sketch with one part per sheet. For information on creating a profile sketch with multiple profiles per sheet, see *Create a multiple profile sketch drawing* (on page 147).

- 1. In the **Console**, right-click the root level of the hierarchy and select **New** on the shortcut menu. On the **Add Component** dialog box, select **Folder** and click **OK**.
 - TIP You could also select an existing folder in the hierarchy and skip this step.
- Right-click the folder and select New. On the Add Component dialog box, go to the Manufacturing tab. Select a Profile Sketch drawing component. For example, select the Mfg Profile Sketch component and click OK. The New Mfg Profile Sketch component is added to the folder. This is the component you use to create your Profile Sketch drawings.
- 3. Right-click the new component and select **Setup** on the shortcut menu. The **Setup** dialog box displays.
- On the General tab, note the View Builder definition. For the delivered Mfg Profile Sketch component, one View and one Report are already defined.



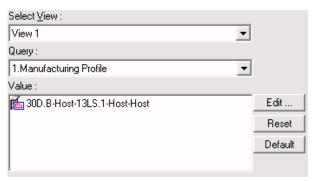
- 5. You can select a view or report and click **Properties** to display the **View Definition** dialog box showing the current property settings for the view or report. Change the properties as needed for the profile sketch drawings you are creating. For more information, see *General Tab* (Setup Dialog Box Drawings by Rule) (on page 108).
- 6. Go to the **Queries** tab to set up the query definition for the profile sketch drawings. The query definition identifies the objects in the model that will be included in the drawing. For more information, see *Queries Tab (Setup Dialog Box Drawings by Rule)* (on page 115).
- 7. In the **Select view type** dropdown, select the view type to which you want the query associated. You can define separate queries per view type or you can use a default query for multiple view types.

Specify a filter to identify the objects used in the query. For a profile sketch drawing, you want to select an assembly that contains the manufactured profiles you need for your drawings. For example, the graphic below shows that the **SeqAssy** filter is being used for the **Mfg Profile Sketch** view type.



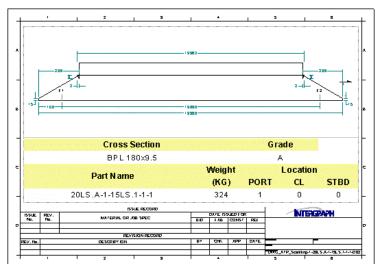
Notice that the **Collation Rule** for the **Mfg Profile Sketch** setup is already set to **CollateToMany**, which means that the drawings will have one part per view.

8. Click **Apply Filter** to run the filter definition. The rest of the dialog updates with additional definition information.



■ NOTE You can further define the view definition using the buttons at the bottom of the **Setup** dialog box. For more information, see *Queries Tab* (Setup Dialog Box - Drawings by Rule) (on page 115).

- 9. Click **OK** on the **Setup** dialog box to save the view and guery definitions.
- 10. If you need to edit the template associated with the drawing component, right click the component and select **Edit Template**. The template associated with the component opens in SmartSketch Drawing Editor so you can edit the layout and presentation for the drawings you are creating. For more information, see *Edit Template (Drawings by Rule)* (on page 128).
- 11. After you complete setup of the Profile Sketch component, right-click the component and select **Create Drawings** to create the drawing documents. The new documents display. The red X icon indicates that the drawings are out-of-date (the current model information needs to be updated in the drawing views).
- 12. Right-click the component again and select **Update** to update the drawing documents with the information from the model. A dialog box shows the update progress.
- 13. When the process is complete, review the information in the dialog box for any problems that may have occurred, then click **OK** to close the dialog box.



14. Right-click the document and select Edit to open it in SmartSketch Drawing Editor.

■ NOTE If you need to create a customized Manufacturing drawing component, create a Drawings by Rule component and define it as a Manufacturing drawing component. For more information, see *Drawings* by *Rule* (on page 102).

Create a multiple profile sketch drawing

The following procedure steps you through creating a multiple profile sketch drawing. This procedure shows how to create a profile sketch with multiple profiles per sheet. For information on creating a profile sketch with one part per sheet, see *Create a profile sketch drawing* (on page 145).

The example shown in this procedure demonstrates:

- More than one profile part is defined for the sketch
- The sketch contains variables defined in a report table
- The report table contains information specific to each profile
 - NOTE The profiles used in a multiple profile sketch drawing must follow certain criteria in order to be shown on the same sketch. For example, they must have the same end cuts and the same profile section type and size.
- 1. In the **Console**, right-click the root level of the hierarchy and select **New** on the shortcut menu. On the **Add Component** dialog box, select **Folder** and click **OK**.
 - TIP You could also select an existing folder in the hierarchy and skip this step.
- Right-click the folder and select New. On the Add Component dialog box, go to the Manufacturing tab. Select a Multiple Profile Sketch component. For example, you could use the Mfg Profile Sketch (Multiple) component and click OK. The New Mfg Profile Sketch (Multiple) component is added to the folder. This is the component you use to create your Multiple Profile Sketch drawings.
- 3. Right-click the new component and select **Setup** on the shortcut menu. The **Setup** dialog box displays.

4. On the **General** tab, note the **View Builder** definition. For the delivered **Mfg Profile Sketch** (**Multiple**) component, one **View** and one **Report** are already defined.



- 5. You can select a view or report and click **Properties** of to display the **View Definition** dialog box showing the current property settings for the view or report. Change the properties as needed for the drawings you are creating. For more information, see *General Tab (Setup Dialog Box Drawings by Rule)* (on page 108).
- 6. Go to the **Queries** tab to set up the query definition for the profile sketch drawings. The query definition identifies the objects in the model that will be included in the drawing. For more information, see *Queries Tab (Setup Dialog Box Drawings by Rule)* (on page 115).
- 7. In the **Select view type** dropdown, select the view type to which you want the query associated. You can define separate queries per view type or you can use a default query for multiple view types.

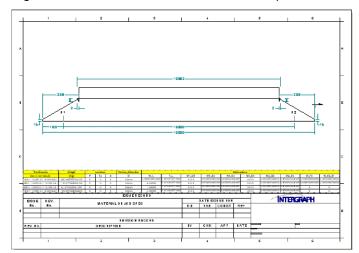
Specify a filter to identify the objects used in the query. For a profile sketch drawing, you want to select an assembly that contains the manufactured profiles you need for your drawings. For example, the graphic below shows that the **SeqAssy** filter is being used for the **Mfg Profile Sketch (Multiple)** view type.



Notice that the **Collation Rule** for the **Mfg Profile Sketch (Multiple)** setup is already set to **CollateSimilarProfiles**, which means that the drawings will have multiple profiles per view.

- 8. Click **Apply Filter** to run the filter definition. The rest of the dialog updates with additional definition information.
 - NOTE You can further define the view definition using the buttons at the bottom of the **Setup** dialog box. For more information, see *Queries Tab* (Setup Dialog Box Drawings by Rule) (on page 115).
- 9. Click **OK** on the **Setup** dialog box to save the view and query definitions.
- 10. If you need to edit the template associated with the drawing component, right click the component and select **Edit Template**. The template associated with the component opens in SmartSketch Drawing Editor so you can edit the layout and presentation for the drawings you are creating. For more information, see *Edit Template (Drawings by Rule)* (on page 128).
- 11. After you complete setup of the Multiple Profile Sketch component, right-click the component and select **Create Drawings** to create the drawing documents. The new documents display. The red X icon indicates that the drawings are out-of-date (the current model information needs to be updated in the drawing views).
- 12. Right-click the component again and select **Update** to update the drawing documents with the information from the model. A dialog box shows the update progress.

- 13. When the process is complete, review the information in the dialog box for any problems that may have occurred, then click **OK** to close the dialog box.
- 14. Right-click the document and select Edit to open it in SmartSketch Drawing Editor.



■ NOTE If you need to create a customized Manufacturing drawing component, create a Drawings by Rule component and define it as a Manufacturing drawing component. For more information, see *Drawings* by *Rule* (on page 102).

Create a pin jig drawing

The following procedure steps you through creating a pin jig drawing. This procedure shows how to use the delivered Mfg PinJig component to create a pin jig drawing with two views, including a remarking view, and a pin height report.

- 1. In the **Console**, right-click the root level of the hierarchy and select **New** on the shortcut menu. On the **Add Component** dialog box, select **Folder** and click **OK**.
 - TIP You could also select an existing folder in the hierarchy and skip this step.
- Right-click the folder and select New. On the Add Component dialog box, go to the Manufacturing tab. Select a PinJig drawing component. For example, select the Mfg PinJig component and click OK. The New Mfg PinJig component is added to the folder. This is the component you use to create your PinJig drawings.
- 3. Right-click the new component and select **Setup** on the shortcut menu. The **Setup** dialog box displays.

4. On the **General** tab, note the **View Builder** definition. For the delivered **Mfg PinJig** component, two **Views** and one **Report** are already defined.



- 5. You can select a view or report and click **Properties** of to display the **View Definition** dialog box showing the current property settings for the view or report. Change the properties as needed for the PinJig drawings you are creating. For more information, see *General Tab (Setup Dialog Box Drawings by Rule)* (on page 108).
- 6. Go to the **Queries** tab to set up the query definition for the PinJig drawings. The query definition identifies the objects in the model that will be included in the drawing. For more information, see *Queries Tab (Setup Dialog Box Drawings by Rule)* (on page 115).
- 7. In the **Select view type** dropdown, select the view type to which you want the query associated. You can define separate queries per view type or you can use a default query for multiple view types.

Specify a filter to identify the objects used in the query. For example, the graphic below shows that the **Pin Jigs** filter is being used for the **Mfg PinJig** view type.



Notice that the **Collation Rule** for the **Mfg PinJig** setup is already set to **CollateToMany**, which means that the drawings will have one part per view.

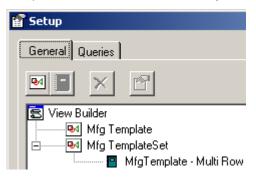
- 8. Click **Apply Filter** to run the filter definition. The rest of the dialog updates with additional definition information.
 - **NOTE** You can further define the view definition using the buttons at the bottom of the **Setup** dialog box. For more information, see *Queries Tab* (Setup Dialog Box Drawings by Rule) (on page 115).
- 9. Click **OK** on the **Setup** dialog box to save the view and query definitions.
- 10. If you need to edit the template associated with the drawing component, right click the component and select **Edit Template**. The template associated with the component opens in SmartSketch Drawing Editor so you can edit the layout and presentation for the drawings you are creating. For more information, see *Edit Template (Drawings by Rule)* (on page 128).

- 11. After you complete setup of the Mfg PinJig component, right-click the component and select **Create Drawings** to create the drawing documents. The new documents display. The red X icon indicates that the drawings are out-of-date (the current model information needs to be updated in the drawing views).
- 12. Right-click the component again and select **Update** to update the drawing documents with the information from the model. A dialog box shows the update progress.
- 13. When the process is complete, review the information in the dialog box for any problems that may have occurred, then click **OK** to close the dialog box.
- 14. Right-click the document and select Edit to open it in SmartSketch Drawing Editor.
- NOTE If you need to create a customized Manufacturing drawing component, create a Drawings by Rule component and define it as a Manufacturing drawing component. For more information, see *Drawings* by *Rule* (on page 102).

Create a template set drawing

The following procedure steps you through creating a template set drawing. This procedure shows how to use the delivered Mfg Template Set component to create a Template Set drawing with two views and a multi-row report.

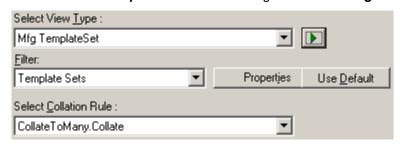
- 1. In the **Console**, right-click the root level of the hierarchy and select **New** on the shortcut menu. On the **Add Component** dialog box, select **Folder** and click **OK**.
 - TIP You could also select an existing folder in the hierarchy and skip this step.
- Right-click the folder and select New. On the Add Component dialog box, go to the Manufacturing tab. Select a Template Set drawing component. For example, select the Mfg Template Set component and click OK. The New Mfg Template Set component is added to the folder. This is the component you use to create your template set drawings.
- 3. Right-click the new component and select **Setup** on the shortcut menu. The **Setup** dialog box displays.
- 4. On the **General** tab, note the **View Builder** definition. For the delivered **Mfg Template Set** component, two **Views** and one **Report** are already defined.



- 5. You can select a view or report and click **Properties** of to display the **View Definition** dialog box showing the current property settings for the view or report. Change the properties as needed for the template set drawings you are creating. For more information, see *General Tab (Setup Dialog Box Drawings by Rule)* (on page 108).
- 6. Go to the **Queries** tab to set up the query definition for the template set drawings. The query definition identifies the objects in the model that will be included in the drawing. For more information, see *Queries Tab (Setup Dialog Box Drawings by Rule)* (on page 115).

7. In the **Select view type** dropdown, select the view type to which you want the query associated. You can define separate queries per view type or you can use a default query for multiple view types.

Specify a filter to identify the objects used in the query. For example, the graphic below shows that the **Template Sets** filter is being used for the **Mfg Template Set** view type.

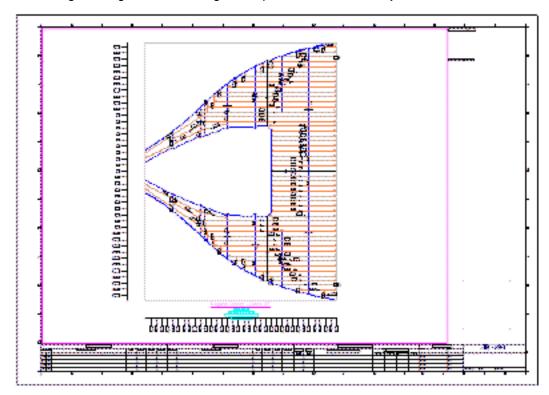


Notice that the **Collation Rule** for the **Mfg Template Set** setup is already set to **CollateToMany**, which means that the drawings will have one part per view.

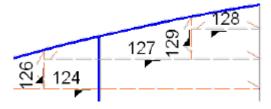
- 8. Click **Apply Filter** to run the filter definition. The rest of the dialog updates with additional definition information.
 - NOTE You can further define the view definition using the buttons at the bottom of the **Setup** dialog box. For more information, see *Queries Tab* (Setup Dialog Box Drawings by Rule) (on page 115).
- 9. Click **OK** on the **Setup** dialog box to save the view and query definitions.
- 10. If you need to edit the template associated with the drawing component, right click the component and select **Edit Template**. The template associated with the component opens in SmartSketch Drawing Editor so you can edit the layout and presentation for the drawings you are creating. For more information, see *Edit Template (Drawings by Rule)* (on page 128).
- 11. After you complete setup of the Mfg Template Set component, right-click the component and select **Create Drawings** to create the drawing documents. The new documents display. The red X icon indicates that the drawings are out-of-date (the current model information needs to be updated in the drawing views).
- 12. Right-click the component again and select **Update** to update the drawing documents with the information from the model. A dialog box shows the update progress.
- 13. When the process is complete, review the information in the dialog box for any problems that may have occurred, then click **OK** to close the dialog box.
- 14. Right-click the document and select **Edit** to open it in SmartSketch Drawing Editor.
- **NOTE** If you need to create a customized Manufacturing drawing component, create a Drawings by Rule component and define it as a Manufacturing drawing component. For more information, see *Drawings* by *Rule* (on page 102).

Scantling Drawings

A scantling drawing is a 2D drawing that represents structural objects.



Objects are represented on the scantling drawing with a standard set of symbols. For example, a profile stiffener is typically resymbolized by a line with an arrow at each end.



What do you want to do?

- Create automated major views for steel order scantling drawings (on page 154)
- Place a manual view by selecting parts (on page 155)
- Place a manual view by selecting a reference plane or offset (on page 157)
- Create a shell profile steel order scantling drawing (on page 158)
- Create weld symbol with double bevel and fillet (on page 161)
- Edit a scantling drawing (on page 162)
- Update a manual view (on page 163)

Create automated major views for steel order scantling drawings

The following procedure creates major views (decks, transverse bulkheads, and longitudinal bulkheads) for a selected block or assembly, using a steel order component delivered with Drawings by Rule. The component includes an asking filter allowing you to select the block or assembly. The workflows are performed in the Drawings and Reports task or from **Tools** > **Drawing Console** in other tasks.

Create a drawing component

 To create a new folder, right-click the drawing root in the Management Console and select New.

The Add Component dialog box displays.

- 2. Select Folder on the General tab of the Add Component dialog box.
- 3. Click OK.
- 4. Right-click the folder, and select **Rename**. Give the folder the needed name.
- 5. Right-click the folder and select **New**.
 - The Add Component dialog box displays.
- 6. Select the **Ship Structure** tab of the **Add Component** dialog box.
- 7. Select the Steel Order by Block or Assembly a or the Steel Order by Block or Assembly (Generic) package and click OK.

The **Steel Order by Block or Assembly** component displays in the Management Console. This component contains views for each of the view types to be created: **Decks - Main Views**, **Longitudinals - Main Views**, **Transversals - Main Views**, **Shell Expansions** and **Shell Profiles**.

8. Right-click the component in the Management Console, and select **Rename**. Give the component the needed name.

Set up and run queries (optional)

- ★ IMPORTANT This procedure can be skipped if you are using the default settings for the component. Perform these steps only when you need to change the queries in the component.
- 1. Right-click the component, and select **Setup**.
 - The **Setup** dialog box displays. For more information, see Setup Dialog Box (Drawings by Rule Component) (on page 108). The **View Builder** hierarchy contains views for each of the view types to be created.
- Select the Queries tab in the Setup dialog box.
- Select All in the Select View Type box.
- An existing asking filter for the component displays in the Filter box. Select another value if needed.
- A collation rule name displays in the Select Collation Rule box. Select another value if needed.

6. Click Ask Filter Inputs *.

The Filter Properties for Asking Filter dialog box displays.

- 7. Select the needed system, part, assembly object, or reference plane from the hierarchy:
 - For a scantling drawing, select a structural part, structural system, or reference plane.
 - For an assembly drawing, select an assembly, block, or assemblyblock.
 - For a piping drawing, select a pipeline.
 - For a pipe support drawing, select a pipeline, or individual hangers and supports.

To include child objects, select Include nested objects. Click OK.

- 8. Click Run Query ▶.
- 9. Select a view in the **Select View Type** box.

Results for the view type display at the bottom of the **Queries** tab for **Views**, **Queries**, and **Values**.

- 10. Each result for **Views** is a view based on the view styles in the component and the object types defined in **Values**.
- 11. If the parts are not the expected results, select a different filter in the **Filter** box, click **Ask Filter Inputs** to select new inputs, and again click **Run Query**.
- 12. Repeat for remaining views in the Select View Type box.
- 13. Click **OK**.

Create drawings

- 1. In the Management Console, right-click the component, and select **Update**.
 - Because you have not selected what is to be displayed on the drawings, the **Filter Properties for Asking Filter** dialog box displays.
- 2. Select a block, assembly, or assemblyblock from the assembly hierarchy. To include child blocks or assemblies, select **Include nested objects.** Click **OK**.
 - The **Drawing Generation** dialog box displays.
- 3. When the **Generation Status** field displays "Creating Documents Complete," click **Exit**.

 One drawing for each selected assembly displays in the Detail View. The drawings are

up-to-date and ✓ is superimposed on each drawing icon.

TIPS

- If a batch server is set up by your system administrator, batch update commands are also available. For more information, see *Batch Commands* (on page 37).
- When a drawing are out-of-date, X is superimposed on the drawing icon.

Place a manual view by selecting parts

The following workflow places a major view (such as the view for a deck, transverse bulkhead, or longitudinal bulkhead) in Drawings by Rule by selecting detailed parts, systems, or leaf systems. The workflows are performed in the Drawings and Reports task or from **Tools** > **Drawing Console** in other tasks.

In this workflow, you directly select detailed parts to associate with the view.

- In Smart 3D, switch to a task other than Drawings and Reports, such as Molded Forms or Structural Detailing.
- 2. Select Tools > Drawing Console.

The **Drawing Console** dialog box displays.

3. Right click a drawing document and select Edit.

SmartSketch Drawing Editor displays. In the Drawings View Explorer, a single sheet displays as a child of the drawing document.

4. In SmartSketch Drawing Editor, click the **Place View** accommand. Click and drag in the drawing area to place a graphic view.

The Drawing View Properties dialog box displays.

5. On the View tab, select More for the Style property.

The Select View Style dialog box displays.

6. Navigate through the hierarchy to **Ruleset Styles**. Select an appropriate steel order ruleset view style, such as **Steel Order (Decks)** or **Steel Order (Generic)**, and click **OK**.

Additional properties display in the View tab of the Drawing View Properties dialog box.

- 7. On the **View** tab, type a value for **Name**.
- 8. On the View tab, select the needed values for Coordinate System, Scale Family, and User Selected Scale.
- 9. On the View tab, select Scantlings for Plate / Profile as the Orientation Rule value.
- 10. Click **OK**.

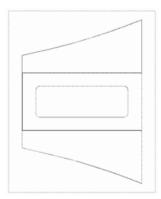
In the Drawings View Explorer, the view displays as a child of the drawing sheet.

- 11. Select the graphic view in the drawing area and click Associate Objects to Views 3.
- 12. Select the 3D application window to make it the active window.

The Associate ribbon displays in the 3D window.

- 13. To associate parts, select **1. Structural Parts or Plane** as the value for **Query** on the **Associate** ribbon.
- 14. In the Workspace Explorer, select plates to associate to the view. Select detailed parts, light (non-detailed) parts, systems, or leaf systems.
- 15. Click Finish on the Associate ribbon to complete the association to the drawing view.

In SmartSketch Drawing Editor, preview geometry of the selected plate parts displays.



16. Update the view. For more information, see *Update a manual view* (on page 163).

Place a manual view by selecting a reference plane or offset

In this workflow, you select a grid plane or an offset from a grid plane to associate parts with the view and select a block, assembly block or assembly to clip the plane selected. You can use the workflow to place a major view, such as the view for a deck, transverse bulkhead, or longitudinal bulkhead.

- 1. In Smart 3D, switch to a task other than Drawings and Reports, such as Molded Forms or Structural Detailing.
- 2. Select Tools > Drawing Console.

The **Drawing Console** dialog box displays.

3. Right click a drawing document and select Edit.

SmartSketch Drawing Editor displays. In the Drawings View Explorer, a single sheet displays as a child of the drawing document.

4. In SmartSketch Drawing Editor, click the **Place View** acommand. Click and drag in the drawing area to place a graphic view.

The **Drawing View Properties** dialog box displays.

5. On the **View** tab, select **More** for the **Style** property.

The **Select View Style** dialog box displays.

6. Navigate through the hierarchy to **Ruleset Styles**. Select an appropriate steel order ruleset view style, such as **Steel Order (Reference Plane)** or **Steel Order - Reference Plane** (**Generic**), and click **OK**.

Additional properties display in the View tab of the Drawing View Properties dialog box.

- 7. On the **View** tab, type a value for **Name**.
- 8. On the View tab, select the needed values for Coordinate System, Scale Family, and User Selected Scale.
- 9. On the View tab, select Scantlings for Plate / Profile as the Orientation Rule value.
- 10. Click **OK**.

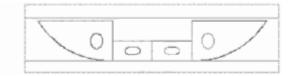
In the Drawings View Explorer, the view \(\bigsim \text{\text{displays}}\) displays as a child of the drawing sheet.

- 11. Select the graphic view in the drawing area and click Associate Objects to Views 3.
- 12. Select the 3D application window to make it the active window.

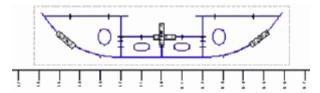
The Associate ribbon displays in the 3D window.

- 13. To associate parts by grid plane, select 1. Structural Parts or Plane as the value for Query on the Associate ribbon. In the Workspace Explorer, select a reference plane coincident with the needed plate parts. For the example in this workflow, a transverse reference plane is selected.
- 14. To associate parts by an offset from a grid plane, select 7. Offset (Optional) as the value for Query on the Associate ribbon. In the Workspace Explorer, select a reference plane. Type an offset value in Value on the Associate ribbon.
- 15. Select plate parts to associate to the view. For the example in this workflow, select plate parts.
- 16. Select **2. Block, Volume or Assembly (Optional)** as the value for **Query** on the **Associate** ribbon.
- 17. In the Workspace Explorer, select the **Assembly** tab.
- 18. Select a block, assembly block or assembly that will define the extents of the plane that is associated with the view being created.
- 19. Click Finish on the Associate ribbon to complete the association to the drawing view.

In SmartSketch Drawing Editor, preview geometry of the plate parts displays. Only parts within the selected block or assembly and coincident with the selected reference plane are associated with the view. The block boundaries are also displayed.



20. Update the view. For more information, see Update a manual view (on page 163).



Create a shell profile steel order scantling drawing

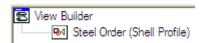
The following procedure creates a **Steel Order (Shell Profiles) Scantling** drawing for a longitudinal shell profile specified by the component filter.

- 1. In the **Drawing Console**, right-click the root level of the hierarchy and select **New** on the shortcut menu. On the **Add Component** dialog box, select **Folder** and click **OK**.
 - TIP You could also select an existing folder in the hierarchy and skip this step.

- Right-click the folder and select New. On the Add Component dialog box, click the Ship Structure tab. Select a Steel Order (Shell Profiles) Scantling drawing component. For example, you could use the delivered Steel Order (Shell Profiles) component and click OK. The New Steel Order (Shell Profiles) component is added to the folder. This is the component you use to create your Scantling drawings.
- 3. Right-click the new component and select **Setup** on the shortcut menu.

The **Setup** dialog box displays.

4. On the **General** tab, note the **View Builder** definition. For the delivered **Steel Order (Shell Profiles)** component, one **View** is already defined.



■ NOTES

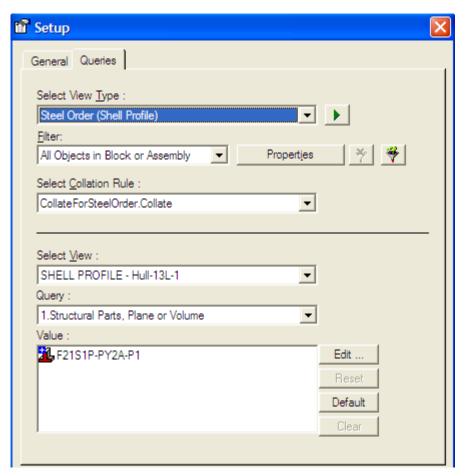
- Add views and reports as needed to define the contents of your drawings.
- Select a view or report and click Properties to display the View Definition dialog box showing the current property settings for the view or report. Change the properties as needed for the shell expansion drawings you are creating. For more information, see General Tab (Setup Dialog Box Drawings by Rule) (on page 108).
- 5. Click the **Queries** tab to set up the query definition for the scantling drawings. The query definition identifies the objects in the model that will be included in the drawing. For more information, see *Queries Tab (Setup Dialog Box Drawings by Rule)* (on page 115).
- 6. In the Select View Type dropdown, select All.
- The asking filter Steel Filter by Block or Assembly is selected in the Filter field. Select another value if needed.
- CollateForSteelOrder.collate is selected in the Select Collation Rule field. Select another value if needed.
- 9. Click Ask Filter Inputs.

The Filter Properties for Asking Filter dialog box displays.

For more information about asking filters, see Select Filter Dialog Box.

- 10. Select a block, assembly, or assemblyblock from the assembly hierarchy and click **OK**.
- 11. Click Run Query.
- 12. Click **OK** on the **Setup** dialog box to save the view and query definitions.
- 13. Select Steel Order (Shell Profiles) in the Select View Type field.

Results for the view type display at the bottom of the **Queries** tab for **Views**, **Queries**, and **Values**. Each result for **Views** is a view with single part or collation of parts as shown in **Values**.

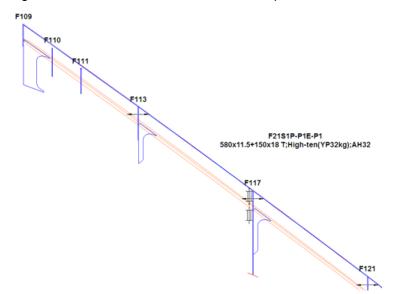


14. The Steel Order (Shell Profiles) package includes a default drawing template. To modify this template, right-click the component and select **Edit Template**.

SmartSketch Drawing Editor opens with the blank drawing template displayed.

You use SmartSketch Drawing Editor to define document, sheet, and region properties before generating the drawing. For additional information on commands available in SmartSketch Drawing Editor, see the SmartSketch Drawing Editor Help.

- 15. After you complete setup of the Steel Order (Shell Profiles) Scantling component, right-click the component and select **Create Drawings**. The new documents display.
 - The red X icon X displays, indicating that the drawings are out-of-date (the current model information needs to be updated in the drawing views).
- 16. Right-click the component again and select **Update** to update the drawing documents with the information from the model.
 - A dialog box shows the update progress.
- 17. When the process is complete, review the information in the dialog box for any problems that may have occurred, then click **Exit** to close the dialog box.



18. Right-click the document and select Edit to open SmartSketch Drawing Editor.

▶ NOTE If you need to create a customized Scantling drawing component, create a Drawings by Rule component and define it as a Scantling drawing component. For more information, see *Drawings by Rule* (on page 102).

Create weld symbol with double bevel and fillet

The following procedure creates a weld symbol that contains double bevels and fillets for use in drawings-by-rule drawing.

- 1. In the Structural Detailing task, select a physical connection that you want to label. You can select **Physical Connections** in the **Locate Filter** field to restrict the **Select** tool to only physical connections.
- Click Properties

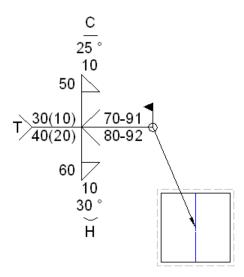
The **StructPhysicalConnection Properties** dialog box displays.

- 3. On the **Parameters** tab, clear the **Rule-Based** options.
- 4. On the **General** tab, select the **Welding** category.
- 5. Modify the values for double bevel and fillet properties as necessary.
- 6. Click OK.

The **StructPhysicalConnection Properties** dialog box closes.

7. Create or update a drawing that contains the physical connections that you modified. For more information on creating rule set drawings, see *Drawings by Rule* (on page 102).

Below is an example of the label that is created for the double bevel and fillet.



NOTE If you manually move the label to the other side of the weld, the label flips to properly indicate the weld information.

Edit a scantling drawing

1. Right-click a drawing document and select Edit.

The drawing opens in SmartSketch Drawing Editor.

- 2. To add an existing view that is not assigned to a drawing, see *Place an Unassigned View* (on page 408).
- 3. To add a section view, see Place a Section View (on page 355).
- 4. To add a detail view, see *Place a Detail View* (on page 359).
- 5. To manually add a drawing view, see *Create a Drawing View* in *Create a new composed drawing* (on page 205).
- 6. To delete a view that is assigned to the drawing, right-click the view In the Drawing View Explorer and select **Delete**.

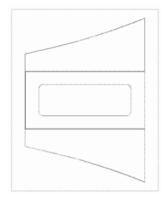
Select **UnAssigned** to move the view to the **Unassigned Folder**, or select **Delete** to delete the view permanently.

- ★ IMPORTANT If you delete a view that is a parent of other views, such as a detail or section view, the Convert or Delete dialog box displays. Select Convert to independent drawing view(s) to save the child view as an independent drawing view, or select Delete to delete the child view.
- 7. In SmartSketch Drawing Editor, click **File > Save**.
- 8. Click File > Exit.

The SmartSketch Drawing Editor window closes.

■ NOTES

Before a view has been updated for the first time, it contains preview geometry in SmartSketch Drawing Editor. For example:



When a view is up-to-date, the view icon looks like this in the Drawing View Explorer: 🔼



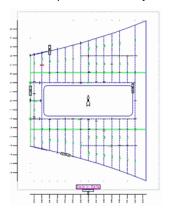
When a view is out-of-date, the view icon looks like this in the Drawing View Explorer: To update an out-of-date drawing, right-click the drawing and select **Update**.



- When a view is unassigned and in the Unassigned Folder of Drawing View Explorer, the view icon looks like this: 💷
- All views in the Unassigned Folder are available for placement on documents created with the **Add Document** command. For more information, see *New* (on page 37).

Update a manual view

- 1. Right-click the out-of-date view in the Drawings View Explorer and select **Update**.
- 2. After the view updates, the view frame in the graphics area shows all geometry defined by the component view style. The view is up-to-date in the Drawing View Explorer.



■ NOTES

You can delete a manual view permanently, or select **UnAssign** to move the view in the **UnAssigned Folder.** If you delete a view that is a parent of other views, such as a detail or section view, the **Convert or Delete** dialog box displays. Select **Convert to independent drawing view(s)** to save the child view as an independent drawing view, or select **Delete** to delete the child view.

- A sheet cannot be deleted unless all views on the sheet are removed first.
- All views within the drawing must be up-to-date or the drawing status will be out-of-date.

For information on the 2D commands available for editing, see the *SmartSketch Drawing Editor Help*.

Filter Properties Dialog Box

Builds a filter or displays the properties of an existing filter. You access this dialog box from the **Select Filter** dialog box, which is available as follows:

- File > Define Workspace Filter option, or Properties button, if a filter is selected.
- Tools > Select by Filter.

This dialog box is entitled **New Filter Properties** or simply **Filter Properties**, depending on whether you are creating a new filter or modifying an existing filter. Its behavior is the same.

The dialog box tabs let you pick the appropriate criteria for the filter. For example, the **System**, **Assembly**, or **Named Space** tabs on the **Filter Properties** dialog box provide for extensive searches, while the **Properties**, **Volume**, **Permission Group**, and **Object Type** tabs assist with more restrictive searches. The **Configuration** tab specifies the permission group assignment of the filter. The **Work Breakdown Structure** (WBS) tab identifies objects in the selected WBS for the filter. For example, you can select projects, contracts, or documents from the WBS.

Name

Specifies the name of the object. If a **Name Rule** is specified, then the software uses that rule to determine this name. If the **Name Rule** value is **User Defined**, then you must type a name in this box.

Asking Filter - user of filter will supply value

Creates an asking filter. An asking filter prompts you for specific values for certain properties.

Include nested objects

Specifies whether you want your search to include all objects under a selected node. For example, when you check this box and then select an object, the software selects all sub-objects under that object. If you do not check this box, you can select objects separately. This option is unavailable for certain tabs on this dialog box.

Lock CTRL key

This option changes the selection mode and allows you to select multiple items across filter tabs without holding down the CTRL key.

Clear All

Removes the object definition. Click **Clear All** if you want to start over and redefine the search criteria.

■ NOTES

When the New dialog box displays, the default is always the last-selected option.

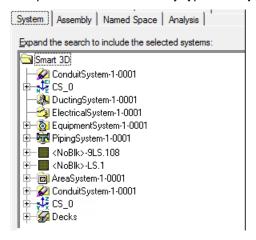
When you double-click a filter on the Select Filter dialog box, the software applies the filter and dismisses the dialog box.

Topics

System Tab (Filter Properties Dialog Box)	165
Assembly Tab (Filter Properties Dialog Box)	166
Named Space Tab (Filter Properties Dialog Box)	166
Analysis Tab (Filter Properties Dialog Box)	167
Work Breakdown Structure Tab (Filter Properties Dialog Box)	167
Permission Group Tab (Filter Properties Dialog Box)	168
Object Type Tab (Filter Properties Dialog Box)	168
Volume Tab (Filter Properties Dialog Box)	169
Properties Tab (Filter Properties Dialog Box)	170
Reference Tab (Filter Properties Dialog Box)	171
Reference 3D Tab (Filter Properties Dialog Box)	171
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Configuration Tab	

System Tab (Filter Properties Dialog Box)

Provides a tree view list of all the available systems you can include in your filter search criteria. A Model is the highest system in the hierarchy and includes all subsystems. Systems can span disciplines and include many types of objects.

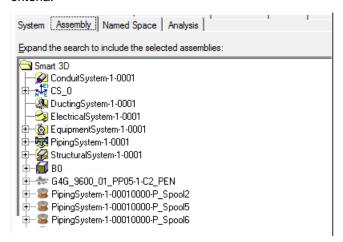


You can select the **Include nested objects** option to specify that you want your search criteria to include all objects within a system. For example, if you select this option, the software selects all children objects when you select a parent system. If you do not select this option, you select only the systems. You can also use the CTRL and SHIFT keys to select multiple objects on this tab.

Nodes with more than 1,000 children display in bold type rather than auto-expanding if some of their children are selected in the filter properties. The selected child nodes highlight when you expand the parent node.

Assembly Tab (Filter Properties Dialog Box)

Provides a tree view list of all the available assemblies you can include in your filter search criteria.



You can select the **Include nested objects** option to specify that you want your search criteria to include all objects within a category. For example, if you select this option, the software selects all objects when you select a category. If you do not select this option, you select the assembly objects separately. You can also use the CTRL and SHIFT keys to select multiple objects on this tab.

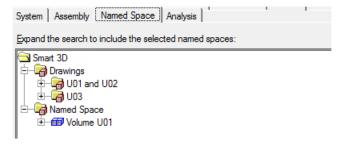
The **Include nested assemblies only** option includes all nested assemblies, assembly blocks, blocks, spools, and penetration spools under the selected assemblies, but not the parts.

■ NOTES

- The Include nested assemblies only option explicitly includes the assemblies and assembly parents that you are working on so that the Refresh Workspace command updates the assembly information without including all of the parts nested under the selected assemblies, such as plate parts, that are not of interest.
- You can only select one of the Include nested objects and Include nested assemblies only options. You can clear both options.

Named Space Tab (Filter Properties Dialog Box)

Provides a list of all the named spaces and drawing volumes you can include in your search.



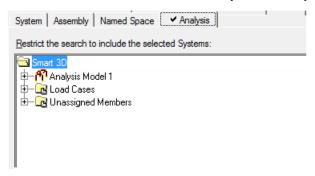
Named spaces are regions in the model, like fire or blast zones. Filtering on named spaces is useful particularly when you work in the Space Management task and need to see the size,

shape, and position of the named spaces that already exist. Drawing volumes are used in the Drawings and Reports task in the drawing creation process.

You can select the **Include nested objects** option to specify that you want your search criteria to include all objects within a category. For example, if you select this option, the software selects all objects when you select a category. If you do not select this option, you select the category and individual objects separately. You can also use the CTRL and SHIFT keys to select multiple objects on this tab.

Analysis Tab (Filter Properties Dialog Box)

Provides a list of all the structural analysis models you can include in your search.



Analysis models are associated with the Structural Analysis task in the software. An analysis model is a non-graphical and logical grouping of member systems that can be sent to a third-party analysis and design solver package.

You can select the **Include nested objects** option to specify that you want your search criteria to include all objects within a category. For example, if you select this option, the software selects all objects when you select a category. If you do not select this option, you select the category and individual objects separately. You can also use the CTRL and SHIFT keys to select multiple objects on this tab.

Work Breakdown Structure Tab (Filter Properties Dialog Box)

Browses a tree view of the model work breakdown structure to include WBS entities only if the **Work Breakdown Structure** tab is used, or restrict the filters to objects assigned to the selected WBS, if other tabs are used.



■ NOTE A simple filter shows only objects assigned to the selected WBS items and the WBS items themselves. To see WBS objects on the Workspace Explorer tab, you must create a compound filter. If you define a filter that contains only one WBS project, this filter returns the WBS project selected on the WBS tab of the Workspace Explorer and any objects assigned to that particular WBS project on the Systems tab in the Workspace Explorer. To see all WBS

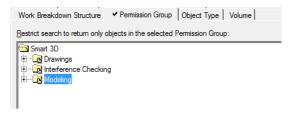
objects on the **WBS** tab in the **Workspace Explorer**, you must create a compound filter. For example, you might create a filter that contains All Systems or WBS Objects. This filter would return all objects on the **Systems** tab and all WBS objects.

The WBS is the breakdown of the Model by the construction work to be performed. The breakdown can consist of the Model at the top level, as well as projects, contracts, and documents. You can modify a property for an object to associate it to a project. You can associate published documents to a contract and then reassign the document from one contract to another. Objects are associated to a document.

You can select the **Include nested objects** option to specify that you want your search criteria to include all objects within a category. For example, if you select this option, the software selects all objects when you select a category. If you do not select this option, you select the category and individual objects separately. You can also use the CTRL and SHIFT keys to select multiple objects on this tab.

Permission Group Tab (Filter Properties Dialog Box)

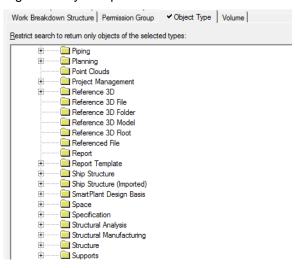
Displays a tree view list of all the permission groups that you can select for your search. The filter selects objects that belong to the groups that you highlight. If you do not highlight any groups, the filter includes all groups in the list.



You can add permission groups in the Project Management task.

Object Type Tab (Filter Properties Dialog Box)

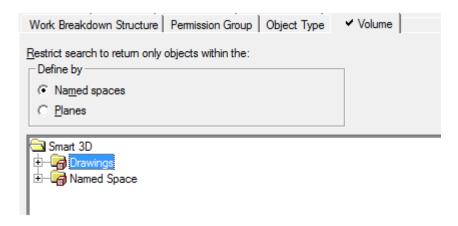
Provides options for you to select specific object types to restrict your filter. The objects are organized by discipline.



This tab provides a list of all the major object types you can include in your search. The filter selects the objects you highlight. If you do not select any objects, the filter includes all objects in the list. To include one or more object types in your filter, press CTRL and click the name of each object type that you want to include.

Volume Tab (Filter Properties Dialog Box)

This tab restricts filter selection to objects within the selected volume and provides two options for defining the volume search method: **Named spaces** or **Planes**. The tree view displays the **Named Spaces** hierarchy or the coordinate system hierarchy depending on the option that you select.



Define by

Named spaces

Displays a tree view of the space hierarchy from which you can choose one or more spaces to include in your search. This option is useful for filtering all objects located within specific spaces. In addition to selecting all the objects inside the specified named spaces, the software retrieves the space itself. You do not need to select the object on the **Named Space** tab as well. To select a particular named space, press CTRL and click as many spaces as you want to include in your search. If you do not select any named spaces, the filter includes all objects in all named spaces.

Planes

Displays a tree view of the reference coordinate system hierarchy in the window, and a group of first and second position coordinate boxes at the bottom. The coordinate system hierarchy is a list of predefined coordinate systems for the model, each having a different origin point. For example, one coordinate system might have an origin point at the corner of a boiler room, another at the center of the building, and so forth.

When you select one of these coordinate systems, the software displays a list of coordinate planes for that system. By selecting a plane and specifying the first and second positions along that plane, your filter selects all objects that fall between the two positions on that plane. The positions automatically appear in the first and second position boxes at the bottom of the dialog box. This option is useful when you want to select objects that are all on a specific level or plane. You can hold CTRL to select the first and second positions in the tree view.

Coordinate system

Specifies a coordinate system. You can define coordinate systems in the Grids task.

1st Position (N, E, EL) or (Y, X, n)

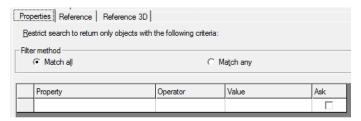
Displays the names of the planes that you select to define the first position of the volume.

2nd Position (N, E, EL) or (Y, X, n)

Displays the names of the planes that you select to define the second position of the volume.

Properties Tab (Filter Properties Dialog Box)

Provides options for selecting object properties that you can use to restrict your search.



Filter Method

You can combine multiple properties on individual rows.

Match All

Returns only those objects matching all of the properties listed in the grid. This method is the same as using the Boolean operator AND.

Match Any

Returns objects matching any property listed in the grid. This method is the same as using the Boolean operator OR.

Property

Lists the properties of objects in the data model in the **Select Properties** dialog box. To select properties and set their data type, select **More** in the field drop-down.

Operator

Select an operator such as <> (not equal) or = (equal).

■ NOTE If you use a wildcard character (*), you must use the **Contains** comparison operator. For example, pumps P-1000A and P-1000B exist in the model. To query for the pumps using properties, select **Match All** and type **Name Contains P***.

Value

Specifies the value of the property.

Ask

Creates an **Asking Filter** that allows you to specify a value for the property when you run the filter. The **Ask** column is so named because the software asks or prompts you to type a value. An administrator or other user with the required permissions establishes the asking

filter and defines a default value. While defining a workspace, you can type a different value for the property. This is not a valid option for Model Data Reuse.

Remove

Removes the selected property from the grid.

Reference Tab (Filter Properties Dialog Box)

Provides a tree view list of the available reference files you can include in your search.



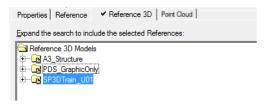
You can also use the CTRL and SHIFT keys to select multiple objects on this tab.

■ NOTES

- To view this tab, you must first insert a file using the Insert > File command.
- When you copy a filter that contains Reference tab information into the Catalog or into a different Model database, the software removes the Reference tab information. Because of this, you cannot create a compound filter that uses Reference tab information. Filters that use Reference tab information are hidden from the tree view on the Compound Filter dialog box. The compound filter ignores any Reference tab information.

Reference 3D Tab (Filter Properties Dialog Box)

Provides a list of all the Reference 3D models you can include in your search.

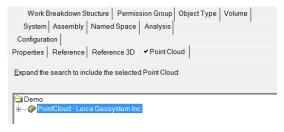


You can also use the CTRL and SHIFT keys to select multiple objects on this tab.

NOTE When you copy a filter that contains Reference 3D tab information into the Catalog or into a different Model database, the software removes the Reference 3D tab information.

Point Cloud (Filter Properties Dialog Box)

Provides a list of all the registered point cloud vendors. You can reference only one point cloud object in your search. You cannot select the parent node to filter the search.



■ NOTE To select point cloud objects for filters, you must install the point cloud vendor software and associate a point cloud model reference with the Model in the Smart 3D Project Management task.

Configuration Tab

Displays the creation, modification, and status information about an object.

NOTE You cannot define the filters using the Configuration tab.

Plant

Displays the name of the model. You cannot change this value.

Permission Group

Specifies the permission group to which the object belongs. You can select another permission group, if needed. Permission groups are created in Project Management.

Transfer

Reassigns ownership of the selected model objects from their current permission group to another satellite or host permission group. This option is only available if the active model or project is replicated in a workshare configuration. The option is not available if all of the objects in the select set already belong to another location and are non-transferable. For more information, see *Transfer Ownership Dialog Box* in the *Common User's Guide*.

NOTE The **Transfer** option does not apply to the filters and surface style rules.

Approval State

Specifies the current status of the selected object or filter. The display depends on your access level. You might be unable to change the status of the object. The list is defined by the ApprovalStatus codelist.

NOTE You can only edit or manipulate an object with a status of **Working**.

Status

Specifies the location of the object in the workflow process. Changing this property sets the **Approval State**. The list is controlled by the ApprovalReason codelist in the ApprovalReason.xls file. You must bulkload this file. For more information, see *ApprovalReason* in the *Reference Data Guide*.

Date Created

Specifies the creation date of the object.

Created by

Specifies the name of the person who created the object.

Date Last Modified

Specifies the date when the object was last modified.

Last Modified by

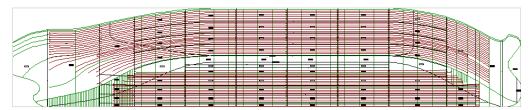
Specifies the name of the person who last modified the object.

Shell Expansion Drawings

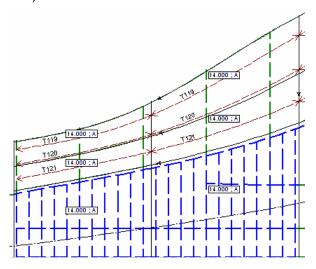
Shell expansion drawings are girth expansions (in the transverse direction) of the hull plate system. The hull plate system is created with the **Imported Plate System** command in the Molded Forms task.

Two types of shell expansion are available:

1. Full port and starboard expansions. For more information, see *Create a shell expansion drawing* (on page 173).



2. Partial expansions, for hull plates that are part of blocks or assemblies. For more information, see *Create automated major views for steel order scantling drawings* (on page 154).



Create a shell expansion drawing

The following procedure steps you through creating shell expansion drawings, using the **Shell Expansion** component delivered with Drawings by Rule. The component creates two drawings: one for the port expansion and one for the starboard expansion. The steps can be performed in the Drawings and Reports task or from **Tools** > **Drawing Console** in other tasks.

Create a Drawing Component

 To create a new folder, right-click the drawing root in the Management Console and select New.

The Add Component dialog box displays.

2. Select Folder on the General tab of the Add Component dialog box.

- 3. Click OK.
- 4. Right-click the folder, and select **Rename**. Give the folder the needed name.
- 5. Right-click the folder and select **New...**. The **Add Component** dialog box displays.
- 6. Select the Ship Structure tab of the Add Component dialog box.
- 7. Select the Shell Expansion package and click OK. The Shell Expansion component displays in the console.
- 8. If needed, right-click the component and select **Rename**. Give the component the needed name.

Set Up the Drawing Component

- Right-click the component and select Setup.... The Setup dialog box displays. For more information, see Setup Dialog Box (Drawings by Rule Component) (on page 108). The View Builder hierarchy contains views for the view types to be created: PORT and STARBOARD.
- 2. Select the **PORT** view and click **Properties** 🖆.
- 3. In the View Definition dialog box, select the needed Coordinate System.
- 4. Click OK.
- 5. Repeat for the STARBOARD view.
- 6. Select the Queries tab in the Setup dialog box.
- Select All in the Select View Type field. The filter Root Hull Plate Systems is selected in the Filter field.
- 8. Click Run Query ▶.
- 9. Select **PORT** in the **Select View Type** field. Results for the view type display at the bottom of the **Queries** tab.
- 10. Repeat for STARBOARD in the Select View Type field.
- 11. Click **OK**.

Create Drawings

- 1. In the Management Console or the Drawing Console, right-click the component and select **Create Drawing(s)**. The **Drawing Generation** dialog box displays.
- 2. When the **Generation Status** field displays "Creating Documents Complete," click **Exit**. Port shell expansion and starboard shell expansion drawings are created. The drawings are out-of-date and a red X icon X is superimposed on each drawing icon.
- 3. To update all drawings, right-click the component in the Management Console and select **Update**. To update individual drawings, right-click drawings and select **Update**. When a drawing is up-to-date, a green check icon ✓ is superimposed on the drawing icon.

See Also

Shell Expansion Drawings (on page 173)

Offshore Drawings

An offshore drawing is a 2D drawing that represents structural and outfitting objects found in an offshore model.

Create a member part drawing

The following procedure creates member part drawings for a selected block or assembly using a drawing-by-query to select the block or assembly. The workflow is performed in the Drawings and Reports task or from **Tools > Drawing Console** in other tasks.

Create a drawing component

 To create a new folder, right-click the drawing root in the Management Console and select New.

The Add Component dialog box displays.

- Select Folder on the General tab of the Add Component dialog box.
- 3. Click OK.
- 4. Right-click the folder, and select **Rename**. Give the folder the needed name.
- 5. Right-click the folder and select New.

The Add Component dialog box displays.

- 6. Select the **Offshore** tab of the **Add Component** dialog box.
- 7. Select the **Member Parts** or the **Member Parts (Generic)** package, and then click

The **Member Part** component displays in the Management Console. This component contains the views to be created: **Looking +Y and Looking -Z**.

8. Right-click the component in the Management Console, and select **Rename**. Give the component the needed name.

Set up and run queries (optional)

- ★IMPORTANT This procedure can be skipped if you are using the default settings for the component. Perform these steps only when you need to change the queries in the component.
- 1. Right-click the component, and select **Setup**.

The **Setup** dialog box displays. For more information, see Setup Dialog Box (Drawings by Rule Component) (on page 108). The **View Builder** hierarchy contains views for each of the view types to be created.

- 2. Select the Queries tab in the Setup dialog box.
- 3. Select All in the Select View Type box.
- 4. An existing asking filter for the component displays in the **Filter** box. Select another value if needed.
- A collation rule name displays in the Select Collation Rule box. Select another value if needed.
- 6. Click Ask Filter Inputs 🔻

The Filter Properties for Asking Filter dialog box displays.

- 7. Select the needed system, part, assembly object, or reference plane from the hierarchy:
 - For a scantling drawing, select a structural part, structural system, or reference plane.
 - For an assembly drawing, select an assembly, block, or assemblyblock.
 - For a piping drawing, select a pipeline.
 - For a pipe support drawing, select a pipeline, or individual hangers and supports.

To include child objects, select Include nested objects. Click OK.

- 8. Click Run Query ▶.
- 9. Select a view in the Select View Type box.

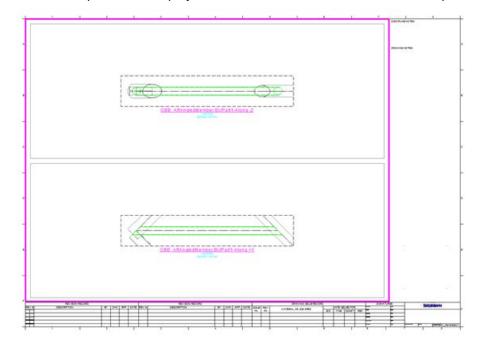
Results for the view type display at the bottom of the **Queries** tab for **Views**, **Queries**, and **Values**.

- 10. Each result for **Views** is a view based on the view styles in the component and the object types defined in **Values**.
- 11. If the parts are not the expected results, select a different filter in the **Filter** box, click **Ask Filter Inputs** * to select new inputs, and again click **Run Query** .
- 12. Repeat for remaining views in the Select View Type box.
- 13. Click **OK**.

Create drawings

- 1. In the Management Console, right-click the component, and select **Update**.
 - Because you have not selected what is to be displayed on the drawings, the **Filter Properties for Asking Filter** dialog box displays.
- From the assembly hierarchy, select one or more blocks, assemblies, or assembly blocks.
 The Drawing Generation dialog box displays.
- 3. When the **Generation Status field** displays "Creating Documents Complete," click **Exit**.

 One drawing for each selected member part displays in the Detail View. The drawings are up-to-date and ✓ is superimposed on each drawing icon.
- 4. Open a drawing.



All member part views display on the same sheet, as shown in the example below.

TIPS

- If a batch server is set up by your system administrator, batch update commands are also available. For more information, see *Batch Commands* (on page 37).
- When a drawing are out-of-date, X is superimposed on the drawing icon.

Create a pipe support drawing

The following procedure creates pipe support drawings for a selected pipeline using a generic drawing-by-rule pipe support component. The workflow is performed in the Drawings and Reports task or from **Tools** > **Drawing Console** in other tasks.

Create a drawing component

 To create a new folder, right-click the drawing root in the Management Console and select New.

The Add Component dialog box displays.

- 2. Select Folder on the General tab of the Add Component dialog box.
- 3. Click OK.
- 4. Right-click the folder, and select **Rename**. Give the folder the needed name.
- 5. Right-click the folder, and select New.

The Add Component dialog box displays.

- 6. Select the Offshore tab of the Add Component dialog box.
- 7. Select the **Pipe Support (Generic)** apackage, and click **OK**.

- The **Pipe Support** component displays in the Management Console. This component contains the views to be created.
- 8. Right-click the component in the Management Console, and select **Rename**. Give the component the needed name.

Set up and run queries (optional)

- ★ IMPORTANT This procedure can be skipped if you are using the default settings for the component. Perform these steps only when you need to change the queries in the component.
- 1. Right-click the component, and select **Setup**.
 - The **Setup** dialog box displays. For more information, see Setup Dialog Box (Drawings by Rule Component) (on page 108). The **View Builder** hierarchy contains views for each of the view types to be created.
- Select the Queries tab in the Setup dialog box.
- 3. Select **All** in the **Select View Type** box.
- 4. An existing asking filter for the component displays in the **Filter** box. Select another value if needed.
- A collation rule name displays in the Select Collation Rule box. Select another value if needed.
- 6. Click Ask Filter Inputs *.

The Filter Properties for Asking Filter dialog box displays.

- 7. Select the needed system, part, assembly object, or reference plane from the hierarchy:
 - For a scantling drawing, select a structural part, structural system, or reference plane.
 - For an assembly drawing, select an assembly, block, or assemblyblock.
 - For a piping drawing, select a pipeline.
 - For a pipe support drawing, select a pipeline, or individual hangers and supports.

To include child objects, select Include nested objects. Click OK.

- Click Run Query ▶.
- 9. Select a view in the **Select View Type** box.

Results for the view type display at the bottom of the **Queries** tab for **Views**, **Queries**, and **Values**.

- 10. Each result for **Views** is a view based on the view styles in the component and the object types defined in **Values**.
- 11. If the parts are not the expected results, select a different filter in the **Filter** box, click **Ask Filter Inputs** * to select new inputs, and again click **Run Query** .
- 12. Repeat for remaining views in the Select View Type box.
- 13. Click **OK**.

Create drawings

1. In the Management Console, right-click the component and select Update.

Because you have not selected what is to be displayed on the drawings, the **Filter Properties for Asking Filter** dialog box displays.

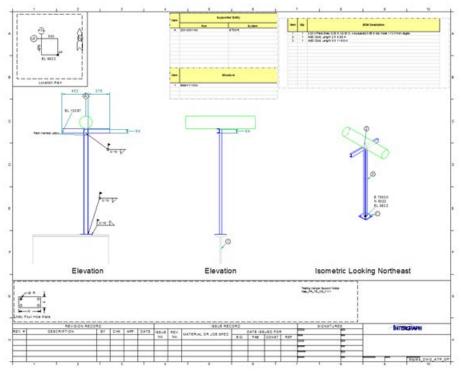
 From the system hierarchy, select a pipeline or a system containing one or more pipe hangers or supports. Select **Include nested objects** to step through the hierarchy and include child hangers and supports. Click **OK**.

The **Drawing Generation** dialog box displays.

- 3. When the **Generation Status** field displays "Creating Documents Complete," click **Exit**.

 One drawing for each selected assembly displays in the Detail View. The drawings are up-to-date and ✓ is superimposed on each drawing icon.
- 4. Open a drawing.

All pipe support view display on the same sheet, as shown in the example below.



TIPS

- If a batch server is set up by your system administrator, batch update commands are also available. For more information, see *Batch Commands* (on page 37).
- To update individual drawings, right-click each drawing, and select **Update**.
- When a drawing are out-of-date, X is superimposed on the drawing icon.

Create a piping drawing

The following procedure creates isometric, plan, and elevation views for a pipeline containing one or more pipe runs, using a generic drawing-by-rule piping component. The workflow is performed in the Drawings and Reports task or from **Tools** > **Drawing Console** in other tasks.

Create a drawing component

 To create a new folder, right-click the drawing root in the Management Console and select New.

The Add Component dialog box displays.

- 2. Select Folder on the General tab of the Add Component dialog box.
- Click OK.
- 4. Right-click the folder, and select **Rename**. Give the folder the needed name.
- 5. Right-click the folder, and select New.

The Add Component dialog box displays.

- 6. Select the **Offshore** tab of the **Add Component** dialog box.
- 7. Select the **Piping (Generic)** apackage, and click **OK**.

The **Piping** component displays in the Management Console. This component contains the views to be created.

8. Right-click the component in the Management Console, and select **Rename**. Give the component the needed name.

Set up and run queries (optional)

- ★ IMPORTANT This procedure can be skipped if you are using the default settings for the component. Perform these steps only when you need to change the queries in the component.
- 1. Right-click the component, and select **Setup**.

The **Setup** dialog box displays. For more information, see Setup Dialog Box (Drawings by Rule Component) (on page 108). The **View Builder** hierarchy contains views for each of the view types to be created.

- 2. Select the Queries tab in the Setup dialog box.
- 3. Select All in the Select View Type box.
- 4. An existing asking filter for the component displays in the **Filter** box. Select another value if needed.
- A collation rule name displays in the Select Collation Rule box. Select another value if needed.
- 6. Click Ask Filter Inputs *.

The Filter Properties for Asking Filter dialog box displays.

- 7. Select the needed system, part, assembly object, or reference plane from the hierarchy:
 - For a scantling drawing, select a structural part, structural system, or reference plane.
 - For an assembly drawing, select an assembly, block, or assemblyblock.
 - For a piping drawing, select a pipeline.
 - For a pipe support drawing, select a pipeline, or individual hangers and supports.

To include child objects, select Include nested objects. Click OK.

8. Click Run Query ▶.

9. Select a view in the Select View Type box.

Results for the view type display at the bottom of the **Queries** tab for **Views**, **Queries**, and **Values**.

- 10. Each result for **Views** is a view based on the view styles in the component and the object types defined in **Values**.
- 11. If the parts are not the expected results, select a different filter in the **Filter** box, click **Ask Filter Inputs** to select new inputs, and again click **Run Query**.
- 12. Repeat for remaining views in the Select View Type box.
- 13. Click **OK**.

Create drawings

- In the Management Console, right-click the component, and select Update.
 Because you have not selected what is to be displayed on the drawings, the Filter Properties for Asking Filter dialog box displays.
- From the system hierarchy, select a pipeline system containing one or more pipe runs. Click OK.

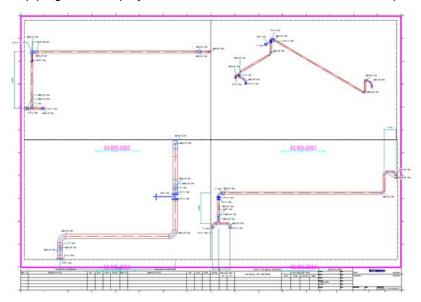
■ NOTE The gathering rule steps through the hierarchy and automatically include child pipe runs, parts, and components.

The **Drawing Generation** dialog box displays.

- 3. When the **Generation Status** field displays "Creating Documents Complete," click **Exit**.

 One drawing for each selected assembly displays in the Detail View. The drawings are up-to-date and ✓ is superimposed on each drawing icon.
- 4. Open a drawing.

All piping views display on the same sheet, as shown in the example below.



TIPS

- If a batch server is set up by your system administrator, batch update commands are also available. For more information, see *Batch Commands* (on page 37).
- To update individual drawings, right-click each drawing, and select **Update**.
- When a drawing is out-of-date, X is superimposed on the drawing icon.

SECTION 9

3D Model by Query

You can create a 3D Model by Query component and base it on a specific filter that determines what objects to document. You can use the 3D Model by Query component to output CAD (SAT) files. The software uses a filter-based query to collect the objects and document them automatically.

Output as Neutral CAD (SAT) Graphics File

You can setup the 3D Model by Query component to output your model objects to a neutral CAD (.sat) file format. Right-click the 3D Model by Query component and select **Setup** to set the filter and navigation rule.

Project Supervisor Setup

Your project supervisor should set up appropriate filters that define the objects to include in the component documents when they are created.

3D Model by Query Component Common Tasks

The following tasks are used to set up a 3D Model by Query component and configure it for exporting CAD (SAT) files.

Define Your Workspace

The defined workspace does not need to include objects being manipulated for export.

Set Up a 3D Model by Query Component

You can create and set up a 3D Model by Query component to create neutral CAD (SAT) graphics files.

Save the 3D Model by Query Component as a Package

Save the 3D Model by Query component as a package so you can join it with a **Drawings by Query Manager** to generate the query drawings. For more information, see *Save a package* (on page 77).

Add a Drawings by Query Manager

Create the **Drawings by Query Manager**, then set it up to specify the "where" portion of the query for the Orthographic, Isogen Isometric Drawing, and 3D Model by Query components. For more information, see *Set up a Drawings by Query Manager component* (on page 186).

Run the Query

Execute the query specified by the 3D Model by Query component and the Query Manager. For more information, see *Run Query (Shortcut Menu)* (on page 70).

Setup (3D Model by Query Component)

Sets options for a 3D Model by Query component. This command is available on the shortcut menu on a 3D Model by Query component.

Setup Dialog Box (3D Model by Query Component) (on page 185)

With this command, you specify a filter that identifies what objects you want collected by the 3D Model by Query component.

(SAT) - The CAD export function supports the following:

- Grouping Selects an equipment object as one object without separately selecting each equipment primitive. Grouping is done on a per object basis.
- Color information and transparency values Retrieves color information and transparency values from the styles and applies them to the SAT file. If no style is selected, the default 3D object style is used. If there is no style associated with an object, green is used.

You do not need to define the workspace or verify that the workspace contains all of the objects required by the 3D Model by Query filter subset.

Set up a 3D Model By Query component

The 3D Model By Query component is used to export model graphics to neutral CAD (.sat) files. Before creating a 3D Model By Query component, you should create filters that specify what objects are collected for your output file.

- 1. Right-click a folder in the Console.
 - TIP To add a folder, right-click the root or another folder.
- On the shortcut menu, select New.

The Add Component dialog box displays.

- Select the 3D Model By Query component on the Add Component dialog box, and click OK.
- 4. Right-click the new 3D Model by Query component, and select Setup.
- 5. Specify a filter to identify what model data to include when the data is exported. Select **More** to display the **Select Filter** dialog box, or click **Properties** to display the current filter properties.
- 6. Select a navigation rule from the **Navigation Rule** drop-down list to include child elements in the document. This is an optional step. For more information on navigation rules, see the *Navigator Rules* section in *Orthographic Drawings* by *Query* (on page 213).
- 7. Click **OK** to save the component settings.

To use the component to generate output files, you need to save it as a package and associate it to a **Drawings by Query Manager**. For more information, see *Set up a Drawings by Query Manager component* (on page 186).

■ NOTES

- To delete a component, right-click it and select **Delete**.
- To rename a component, right-click it and select Rename.

See Also

Setup (3D Model by Query Component) (on page 184)

Setup Dialog Box (3D Model by Query Component)

Sets options for the 3D Model by Query component. You access this dialog box when you right-click a 3D Model by Query component and select **Setup** on the shortcut menu.

Filter - Identifies the filter that defines what objects to include in the query. The filter needs to be specific to the objects that you want to export. The software uses the filter to determine the objects included in the drawings when they are generated. Select **More** in the **Filter** dropdown list to display the **Select Filter** dialog box. Click **Properties** to display the **Filter Properties** dialog box. For more information on the **Select Filter** dialog box, see *Select Filter Dialog Box* (on page 122).

TIPS

- You do not need to define the workspace to contain all of the objects required by the 3D Model by Query filter subset.
- For more information on query filters, see Drawings by Query Filters (on page 216).

Navigation Rule

Specifies the navigation rule to use to include child elements in the document. This is an optional setting. For more information on navigation rules, see the *Navigator Rules* section in *Orthographic Drawings by Query* (on page 213).

Output File Type

Specifies the output file type.

See Also

Setup (3D Model by Query Component) (on page 184)

Setup (Drawings By Query Manager Component)

Sets options for creating a Drawings by Query Manager component for use with a 3D Model by Query package. This command is available on the right-click shortcut menu for Drawings by Query Manager components.

The Drawings by Query Manager component is used in conjunction with other components, such as the 3D Model by Query, Orthographic Drawing by Query, and Isogen Isometric Drawing by Query components, to complete the query for objects in the model. The Drawings by Query Manager provides the filter that specifies the "where" side of the query. It tells the query "where" to look for the objects specified by the component "what" filter.

When using the Drawings by Query Manager component in conjunction with a 3D Model by Query package, additional options for output files are available that allow you to save files to disk.

Setup Dialog Box (Drawings by Query Manager Component) (on page 187)

Set up a Drawings by Query Manager component

The Drawings by Query Manager component works in conjunction with the Orthographic Drawing by Query, Isogen Isometric Drawing by Query, and 3D Model by Query components. Before using this command, you must create packages for your 3D Model by Query component. For more information, see *Save Package Command* (on page 76).

1. Right-click the folder where you want to create your **Drawings by Query Manager**.

The Add Component dialog box displays.

- TIP You can store the **Drawings by Query Manager** anywhere in the **Console**, but it is best to store it in the same location as the components with which it works.
- 2. Select the **Drawings by Query Manager** component, and then click **OK**.

The Drawings by Query Manager component is created in the folder.

Right-click the **Drawings by Query Manager** component and select **Setup** to specify the properties for the component.

The Setup dialog box displays.

- 4. Specify a filter in the **Filter** field. The drop-down shows the most recently selected filters. Select **More** in the drop-down list to display the **Select Filter** dialog box and specify a filter. Click **Properties** to display the current filter properties.
 - **NOTE** The filter you select is the "where" portion of the query, as opposed to the "what" portion specified when you setup the 3D Model by Query component. The filter you specify here tells where in the model you want to look for the objects. For more information on filters for Drawings by Query, see *Drawings by Query Filters* (on page 216).
- In the Package field, specify the 3D Model by Query package you created. The drop-down contains the most recently selected packages. Select More to display the Select Package dialog box.
- 6. Define a path to save the exported files in the **Path to save** field. You can type in an existing folder path, or click the browse button to browse to or create a new folder.
- 7. Click **OK** to save the settings.

To create the drawings, you need to run the query. For more information, see *Run Query (Shortcut Menu)* (on page 70).

■ NOTES

 Because the AutoCAD software can open only SAT files of version 5, the ACIS version in the SAT file is written as 5 by default. If you want to output the current ACIS version, contact Intergraph Support (http://www.intergraph.com/support) or your local office.

See Also

Setup (Drawings by Query Manager Component) (on page 185) Export color and transparency styles (on page 187)

Setup Dialog Box (Drawings By Query Manager Component)

Sets options for creating Drawings by Query Manager components associated with a 3D Model by Query package.

■ NOTE You must create an Orthographic Drawing by Query, Isogen Isometric Drawing by Query, or 3D Model by Query package before setting up the Drawings by Query Manager component. For more information, see Save Package Command (on page 76).

Filter

Identifies the filter to use to define the *where* portion of the query. The software uses the filter to determine where to look for the objects requested in Orthographic Drawing by Query and Isogen Isometric Drawing by Query components when they are generated. Select **More** in the list to display the **Select Filter** dialog box. Click **Properties** to display the current filter properties. For more information on filters for the Drawings by Query Manager, see *Drawings by Query Filters* (on page 216).

Package

Specifies the package to use in completing the query.

Output

Disk only (do not save to database)

Save the output file locally. A copy of the output file is not saved to the database. You cannot change this option.

Path to save

Specify a local folder for the output file. You cannot leave this field blank.

See Also

Setup (3D Model by Query Component) (on page 184)

Export color and transparency styles

If you are exporting objects to a CAD format graphics file, you can also export color and transparency styles to the SAT file when it is created.

1. Right-click a document, and select Properties.

The **Properties** dialog box displays.

- 2. On the Surface Styles tab, select the style, and then click Add.
- 3. Right-click the document component, and select **Update Now**.

The SAT file is created with the selected style.

■ NOTE Style rules on the Surface Styles tab are created in the **Common** task. You can also create new styles by clicking **New**.

SECTION 10

3D Model Data

You can create a 3D Model Data component and base it on a specific filter. The filter defines the contents of the component documents when they are created. You can use the 3D Model Data component to output SmartPlant Review files or CAD (SAT) files.

Output as Neutral CAD (SAT) Graphics File

You can setup the 3D Model Data component to output your model objects to a neutral CAD (.sat) file format. Right-click the 3D Model Data component, and select **Setup** to set the filter and output file path.

Output as SmartPlant Review File

You can setup the 3D Model Data component to output your model objects as SmartPlant Review files depending on how you set up the 3D Model Data component. The software creates .vue and .xml files by default. If you select the **Generate streaming vue output (.zvf file)** option on the **Setup** dialog box, the software creates all three SmartPlant Review files: .zvf, .vue, and .xml.

You can set the output on the 3D Model Data component to save the SmartPlant Review (SPR) file to disk only, database only or both.

- Select Disk only (do not save to database) to save the generated graphics and data files to the specified path. You can then update and publish the 3D Model Data files to SmartPlant Foundation in an integrated environment. This option helps prevent data congestion by allowing you to save and publish your files locally.
- Clear Disk only (do not save to database) and do not specify output paths to save the data only to the database.
- Clear Disk only (do not save to database and specify the output paths to save the data to the database, and as files to the specified paths.

The basic workflow for creating a 3D Model Data component is as follows:

- Create a 3D Model Data component, and set it up with a filter and output file information.
- Create the 3D Model Data document (one document per 3D Model Data component).

■ NOTE SmartPlant Review shows the objects from the .vue file using global coordinates. If you plan to save the 3D Model Data component documents to a .vue file using the Save as SmartPlant Review File command, right-click the component and select Properties to check the Style tab Coordinate System property setting. You want to make sure the Plant Monument Coordinate Offset is passed correctly to SPR when creating the .vue file. The offset value allows you to see the original coordinates relative to the new SPR coordinate system.

- Revise the documents if publishing to SmartPlant Foundation.
- Set properties Surface Styles and Aspects properties as needed on the 3D Model Data documents.

- Update the documents using Update Now or Batch > Update on the 3D Model Data component shortcut menu.
- If SmartPlant Foundation requires a password, you are prompted to type it when updating 3D Model Data documents.
- Save the documents to a predefined location for viewing in SmartPlant Review or publish the documents to the registered SmartPlant Foundation plant.

Recommendations for Exporting to SmartPlant Review

- The number of objects generated by a 3D Model Data component and exported successfully to a SmartPlant Review file depends largely on the type of objects and your hardware resources. We recommend that you limit each 3D Model Data component filter.
- Monitor the error logs regularly for resource issues, even if the specified filter worked initially. You can add more objects to the model meeting the filter criteria.
- SmartPlant Review (SPR) version 6.1.0.15 (or higher) allows you to open multiple .vue files simultaneously. Refer to your SmartPlant Review documentation for more information. When you open .vue and .xml files in SPR for the first time, SPR builds a database containing the tag information for the files. This process can take a significant amount of time.
- SmartPlant Review (SPR) version 6.2.0.29 (or higher) supports turning SmartPlant aspects
 on and off. All aspects are turned on by default in SPR. The SPRSchema.txt file can be
 customized to add any customized aspects.

■ NOTE The 3D View Control used for viewing the published graphics in SmartPlant Markup Plus does not currently support turning aspects on and off.

Project Supervisor Setup

Your project supervisor should set up appropriate filters that define the objects to include in the component documents when they are created.

You must install the **SmartPlant Schema Component** and the **SmartPlant Client** to use this component.

3D Model Data Component Common Tasks

The following tasks are used to set up a 3D Model Data component and configure it for viewing in SmartPlant Review or publishing.

Define Your Workspace

The defined workspace does not need to include objects being manipulated for SmartPlant Review.

Setup a 3D Model Data Component

You can create and set up a 3D Model Data component to create SmartPlant Review output files or neutral CAD (SAT) graphics files.

Create the 3D Model Data Document

To create the 3D Model Data document, right-click the component and select the **Create Drawing** command. When this command is complete, the document is listed in the **Detail View**.

Set Surface Style Rules and Aspects for 3D Model Data Documents

Before you update your 3D Model Data documents, set the surface style rules and aspects to use for the model objects. You can specify the properties for the 3D Model Data component and documents by right-clicking and selecting **Properties**.

Update 3D Model Data Documents

Update your 3D Model Data documents using **Update Now** or **Batch** > **Update** on the component or document shortcut menu.

Save Data as a SmartPlant Review File

You can output your 3D Model Data to a SmartPlant Review (.vue) file whether you are working in a stand-alone or integrated environment.

Set Revision Information

The document revision process is separate from the publishing process, making it possible to revise a document locally and save it to the database without re-publishing the document.

Publish Data

If your model has been registered using the SmartPlant Registration Wizard, you can publish your 3D Model data for retrieval in other tools.

Setup (3D Model Data Component)

Sets options for a 3D Model Data component. This command is available on the shortcut menu on a 3D Model Data component.

Setup Dialog Box (3D Model Data Component) (on page 192)

With this command, you specify a filter that identifies the objects you want collected by the 3D Model Data component. You have several combination options for output of the objects:

- SmartPlant Review graphics file (VUE)
- CAD output file (SAT)

The CAD export function supports the following:

 Color information and transparency values - The software can retrieve color information and transparency values from the styles and apply them to the SAT file. If no style is selected, the default 3D object style is applied. If there is no style associated with an object, green is applied by default.

You do not need to define the workspace or verify that the workspace contains all of the objects required by the 3D Model by Query filter subset.

■ NOTES

You can publish .zvf files and launch SmartPlant Review to view the .zvf files. You can also use File > View and Markup to use SmartPlant Markup Plus to navigate the model. You do not have to use the Save as SmartPlant Review command to use the SmartPlant Review features.

 After you create a component, you must update the documents to extract the data from the database to create the .vue, .zvf, and .xml files.

Setup a 3D Model Data component

The 3D Model Data component is used with the **Save as SmartPlant Review** and **Publish** commands. It is also used to output model graphics to neutral CAD (.sat) files. Before creating a 3D Model Data component, you should create filters that specify the objects required for your output.

- 1. Right-click a folder in the Console.
 - TIP To add a folder, right-click the root or another folder.
- 2. On the shortcut menu, select New.

The Add Component dialog box displays.

- 3. Select the 3D Model Data component, and click OK.
- 4. Right-click the new 3D Model Data component and then select Setup.
- 5. Specify a filter to identify the model data to include when the data is saved or published. Select **More** to display the **Select Filter** dialog box. Click **Properties** to display the current filter properties.
- If you need to output the 3D Model Data to SmartPlant Review, select Generate
 SmartPlant Review output (.vue file). If you need to output the data as a neutral CAD format graphics file (.sat file), select the Generate CAD output (.sat file) option.
- 7. Select or clear the **Disk only (do not save to database)** option to specify how to save the 3D Model Data. If you are registered in an integrated environment, checking this option allows you to update and publish the output files to SmartPlant Foundation.
 - **NOTE** When saving as SmartPlant Review files, the behavior of the 3D Model Data setup is different depending on whether you are registered to work in an integrated environment or not. For more information on how this option affects the saved data, see *Setup Dialog Box (3D Model Data Component)* (on page 192). For more information on registering your model using the SmartPlant Registration Wizard, see the section titled *Working in an Integrated Environment* in the *Intergraph Smart*TM *3D Installation Guide*, available from the **Help > Printable Guides** command.
- 8. Specify file paths for the data file and the graphics file as needed.
- 9. Click **OK** to save the component settings and create the output files as specified.

■ NOTES

- After you create a component, you must update the documents to extract the data from the
 database and create the .vue, .zvf, and .xml files. For more information on tasks associated
 with the 3D Model Data component, see 3D Model Data Component Common Tasks (on
 page 189).
- If you are setting up the 3D Model Data component with the intention of saving it as a SmartPlant Review (SPR) file, right-click the component and select **Properties** and go to the **Style** tab to make sure the **Coordinate System** property is set appropriately so that the **Plant Monument Coordinate Offset** is passed correctly to SPR when creating the VUE file. This is because SPR shows the objects from the VUE file using global coordinates. The offset value allows you to see the original coordinates relative to the new SPR coordinate

- system. For more information on 3D Model Data components, see 3D Model Data (on page 188). For information on saving to SPR, see Save as SmartPlant Review File (on page 417).
- If you are setting up the 3D Model Data component for output to the CAD format graphics file, you can export color and transparency style rules along with the objects to the SAT file when it is created. To set the style for export, right-click the 3D Model Data component and select **Properties**. Go to the **Surface Styles and Aspects** tab, select the style and click **Add**. This property tab includes the style rules that are created in the **Common** task. You can also create new styles by clicking **New**. After the style is set, right-click the 3D Model Data component and select **Update Now** to create the SAT file with the specified style.
- Because the AutoCAD software can open only SAT files of version 5, the ACIS version in the SAT file is written as 5 by default. If you want to output the current ACIS version, contact Intergraph Support (http://www.intergraph.com/support) or your local office.
- For the SAT output, all the bodies are merged and then saved to the SAT file. If you are
 using viewers, such as MicroStation, that cannot open merged bodies, contact Intergraph
 Support (http://www.intergraph.com/support) or your local office.

See Also

Publishing Documents (on page 424)
Setup (3D Model Data Component) (on page 190)
Find Documents to Publish (on page 441)
Publish (on page 427)

Setup Dialog Box (3D Model Data Component)

Sets options for the 3D Model Data component. You access this dialog box when you right-click a 3D Model Data component and select **Setup** on the shortcut menu.

Filter

Identifies the filter to use to define the objects to include in the component definition. The filter needs to be specific to the objects that you want to publish or to save to a SmartPlant Review file.

TIP You do not need to define the workspace to contain all of the objects required by the 3D Model Data filter subset.

Options

Specifies how the 3D Model Data is saved. The setup for your 3D Model Data component is different depending on whether or not you are registered to work in an integrated environment.

Generate SmartPlant Review output (.vue file)

Indicates that you want the 3D Model Data saved as a SmartPlant Review .vue file.

Generate streaming vue output (.zvf file)

Indicates that you want the 3D Model Data saved as a .zvf file in addition to a .vue file. Check this option if you intend to attach this data as a reference to another plant. The .zvf format allows you to attach this plant data as a reference to another plant using Reference 3D functionality.

Generate CAD output (.sat file)

Indicates that you want the 3D Model Data saved as an SAT graphics file.

Disk only (do not save to database)

Specifies how the graphics and/or data files are saved:

- Select Disk only (do not save to database) to save the files only to the specified path.
 You can then update and publish the .vue or .zvf file to SmartPlant Foundation from the designated locations.
- Clear the Disk only (do not save to database) and do not specify output paths for the data to only be saved to the database.
- Clear the Disk only (do not save to database and specify the output paths to save the data to the database and as files to the specified paths.

You can change this option at any time during your work so that you can determine the best save and publish options for your specific environment.

Path to save graphics file

Specifies the file to which the graphics for the 3D Model Data will be saved. Click the ellipsis button to browse for the correct file.

Path to save data file

Specifies the file to which the data will be saved. Click the ellipsis button to browse for the correct file. This box is automatically populated, and the ellipsis button does not display when you specify a .vue file. For example, if you specify C:\temp\tpjtest.vue in the **Path to save graphics file** box, then the **Path to save data file** box automatically displays C:\temp\tpjtest.xml.

■ NOTES

- You can publish .zvf files and launch SmartPlant Review to view the .zvf files from within SmartPlant Foundation. You do not have to use the Save as SmartPlant Review command to use the SmartPlant Review features. For more information on publishing, see Publish (on page 427).
- For more information about the SmartPlant Registration Wizard, see Working in an Integrated Environment in the Intergraph Smart™ 3D Installation Guide, available from Help > Printable Guides.

See Also

Setup (3D Model Data Component) (on page 190) Setup a 3D Model Data component (on page 191) 3D Model Data Component Common Tasks (on page 189)

SECTION 11

Exporting 3D Model Graphics to MicroStation

You can export 3D model graphics directly to MicroStation J (V7) and MicroStation V8 DGN file formats. The actual export is not a drawing file. It is an actual 3D file translated into MicroStation graphic types. While the workflow is similar to that of regular volume drawings, there are some differences.

Create a MicroStation component with the **Add Component** command. After the component is created, right-click and select **Setup** from the shortcut menu. The **Setup** command allows you to define the MicroStation version, seed file, and style to use in generating the component documents.

The view style you specify for the component determines which model object graphics are included in the component documents. The view style also specifies the layers on which the graphics are placed as well as colors and linear styles when imported in MicroStation. The MicroStation version you specify defines the type of DGN file to be generated. The seed file selected determines the MicroStation symbology used within the document.

■ NOTE Layers are referred to as Levels in MicroStation. Levels are defined by integers 1-63 and must be typed in the Layer property in the View Style properties dialog box.

See Also

Create MicroStation DGN files (on page 195) Setup (MicroStation DGN Files) (on page 194)

Setup (MicroStation DGN Files)

Sets options for creating a MicroStation DGN file. This command is available on the shortcut menu on the MicroStation 3D DGN component.

Setup Dialog Box (MicroStation DGN Files) (on page 194)

See Also

Create MicroStation DGN files (on page 195)

Setup Dialog Box (MicroStation DGN Files)

MicroStation Version - Specifies the type of DGN file to be generated. By default, the software selects the **MicroStation V7 DGN** file.

■ NOTE You must install MicroStation V8 XM (08.09) or higher on your computer to generate MicroStation V8 DGN files.

Seed File

Specifies the seed file to use to create the document. Select **More**, navigate to the appropriate folder, and then select the seed file.

■ NOTES

- You must specify the MicroStation version before you specify the seed file. Seed files vary for different versions of DGN file formats.
- Seed files for MicroStation V8 DGN files are located in the MicroStation V8 folder. Seed files for MicroStation V7 DGN files are in the Templates folder.

View Style

Selects the defined view style to use in determining which 3D model objects display in the document. Click **Properties** to display the **View Style Properties** dialog for the selected style. For more information, see "Define View Style Dialog Box" in the *Drawings* and *Reports Reference Data Guide*.

See Also

Create MicroStation DGN files (on page 195) Setup (MicroStation DGN Files) (on page 194)

Create Drawing (MicroStation DGN Files)

Generates a document for the MicroStation 3D DGN components. Before you can use this command to generate documents, you must complete the steps below.

- Create at least one MicroStation 3D DGN component in the Console hierarchy.
- Right-click on the component and select **Setup** to define the MicroStation version, seed file, and view style to use in generating the documents for export into MicroStation.
- Create at least one volume with the Place Drawing Volume commands in the Space Management task.

■ NOTES

- The MicroStation DGN data reflects the clipping applied in 3D application and the style information defined in the graphic rule of the view style. No data is persisted on any of the model objects, but all objects are mapped to levels as specified by the graphic rule. If the graphic rule does not assign a level, the graphics are placed on level 1.
- During update of the documents, the software saves the MicroStation J (V7) or MicroStation V8 DGN files to the model database. Use the Save As command to export the DGN files from the database to another location.

Create MicroStation DGN files

This procedure shows how to create MicroStation 3D DGN documents for export into MicroStation J (V7) and MicroStation V8 DGN files. For more information, see *Exporting 3D Model Graphics to MicroStation* (on page 194).

- 1. Create a MicroStation 3D DGN component.
 - Add a package of components (on page 42)
- 2. Right-click the MicroStation 3D DGN component, and select **Setup** from the shortcut menu.
 - The **Setup** dialog box displays.
- Select the required MicroStation Version DGN file to generate. You can select Generate v7 DGN file or Generate v8 DGN file.

■ NOTES

- Generate v7 DGN file is selected by default.
- Both MicroStation v7 and v8 can reside on the same computer.
- 4. Select the appropriate seed file to use from the options available.
- 5. Select the view style you want to use to determine what objects are displayed in the document.
- 6. Create one or more drawing volumes to associate with the drawing view by switching to the Space Management task and clicking one of the **Place Drawing Volume** commands.
- 7. Right-click the MicroStation 3D DGN drawings component, and select **Create Drawing(s)** to create the blank drawing.
- 8. Right-click on the document, and click **Update Now** to update the information displayed in the drawing.
 - TIP You must place a Drawing type volume from the Space Management task before you can update the drawing.
- 9. You can open the DGN file in MicroStation after it is updated. Select **Open** and the drawing component.
- 10. You can export DGN files by right-clicking a folder, component, or single document, and then selecting **Save As** on the shortcut menu.

■ NOTES

The delivered seed files contain master units as m (meters) and feet ("). When exporting graphics to MicroStation Version v7 DGN file format, make sure the selected MicroStation Version v7 DGN seed file has one of the following labels as working units.

Working Units Labels	Working Unit Name
um	Micrometer
mm	Millimeter
cm	Centimeter
dm	Decimeter
m	Meter
hm	Hectometer
km	Kilometer
ui	MicroInch
in, "	Inch
ft, '	Foot
yd	Yard

mi Mile

- The size and measurement of the graphics is accurate only if the units present in the seed file are within the set of units listed in the previous table. Otherwise, the working units of the seed file are treated as meters. These labels are not case-sensitive.
- The MicroStation DGN data reflects the clipping applied in 3D application and the style information defined in the graphic rule of the view style. No data is persisted on any of the model objects, but all objects are mapped to levels as specified by the graphic rule. If the graphic rule does not assign a level, the graphics are placed on level 1.
- After a view style is selected on the **Setup** dialog box, you can use the view style properties
 to add, edit, or remove filters that determine what objects are included in the document.
- The Layer property in a graphic rule assigns objects to a DGN level. For the purposes of saving to a 3D DGN file, the software only recognizes levels 1 to 63. Other values, or an empty field, are interpreted as level 1.

SECTION 12

Generic Module Folder

The Generic Module Folder component provides a way for you to run your custom VB modules to create custom drawings. You set up the Generic Module Folder component to use your custom VB module. For example, you might create a VB module to create MicroStation 3D drawings, AutoCAD 3D drawings, or Electrical Isometric drawings. The Generic Module Folder is simply a container for your VB modules. You create the component just like other components, but when you perform **Setup** on the component, you specify a custom VB module.

The software builds the list of available VB modules from the SharedContent library, specifically from [Reference Data Folder]\SharedContent\Drawings\Catalog\DwgTypeModules. You must store your VB module in this location to have access to it when setting up the component.

Custom VB Module

The behavior of the component depends entirely on how the VB module is developed. Everything is controlled by the VB module. When you right-click the component, the VB module determines the menu items that are available. If the module has a command that creates documents, you see the **Create Documents** command. If, for whatever reason, the module does not need documents, the document-related commands do not display on the shortcut menu. Other commands the VB module can include are **Rename**, **Delete**, **Refresh**, and **Print**. For more information on general commands available, see *Shortcut Menus* in any of the Smart 3D Drawings user's guides, available from **Help > Printable Guides**.

If the module supports publishing, the right-click menu includes the **Publish** command. This command is available only if your model has been registered using the Smart 3D Registration Wizard. For more information, see *Publishing Documents* in the *Integration Reference Guide*, or in any of the Smart 3D Drawings user's guides, available from **Help > Printable Guides**.

For information on the interfaces used to create a custom VB drawing module, see the Intergraph SmartTM 3D Programmer's Guide. Contact your administrator or Intergraph Support if you need the Intergraph SmartTM 3D Programmer's Guide. You can find support information on our web site http://support.intergraph.com (http://support.intergraph.com/).

See Also

Use a Generic Module Folder component (on page 199) Setup (Generic Module Folder Component) (on page 198)

Setup (Generic Module Folder Component)

Sets the custom VB module associated with the Generic Module Folder component. This command is available on the right-click popup menu for the Generic Module Folder component.

Setup Dialog Box (Generic Module Folder Component) (on page 199)

Select Module Dialog Box (Generic Module Folder Component) (on page 199)

Use a Generic Module Folder component

The Generic Module Folder is a container component for a custom VB module. Before you perform this procedure, ensure that your created custom VB modules are stored in \Symbols\Drawings\Catalog\DwgTypeModules.

 Right-click the folder in the hierarchy where you want to store the Generic Module Folder component, then select **More** from the **New** menu.

The Add Component dialog box displays.

2. Select the Generic Module Folder component, then click OK.

The Generic Module Folder component is created in the folder.

3. Right-click the Generic Module Folder component and select **Setup**.

The Setup dialog box displays.

- Specify a custom VB module from the dropdown list. Click More to display the Select Module dialog box, which provides a hierarchy of modules available from \Symbols\Drawings\Catalog\DwgTypeModules.
- 5. Click **OK** to save the changes to the Generic Module Folder component.

The new custom component is added to the selected folder in the hierarchy.

Right-click the new custom component and use the available commands to perform operations on the component. The commands available depend entirely on the functionality provided in the VB module.

See Also

Setup (Generic Module Folder Component) (on page 198) Setup Dialog Box (Generic Module Folder Component) (on page 199) Select Module Dialog Box (Generic Module Folder Component) (on page 199)

Setup Dialog Box (Generic Module Folder Component)

Sets the custom VB module for the Generic Module Folder component.

Modules

Specifies a custom VB module. The dropdown list contains a list of the most recently used VB modules. Click **More** to display the **Select Module** dialog box. For more information, see Select Module Dialog Box (Generic Module Folder Component) (on page 199).

See Also

Setup (Generic Module Folder Component) (on page 198) Use a Generic Module Folder component (on page 199)

Select Module Dialog Box (Generic Module Folder Component)

Provides a hierarchical list of available VB modules. The hierarchy includes all VB modules located in \Symbols\Drawings\Catalog\DwgTypeModules. This dialog box displays when you select **More** in the **Modules** dropdown on the **Setup** dialog for a Generic Module Folder component.

See Also

Setup (Generic Module Folder Component) (on page 198) Use a Generic Module Folder component (on page 199) Setup Dialog Box (Generic Module Folder Component) (on page 199)

SECTION 13

Composed Drawings

Composed drawings are orthographic drawings created in a 3D task, such as Common, through the Drawing Console. The composed drawing component manages the composed drawings you create. Composed drawings are flexible, allowing you to have views that are managed by a drawing region and to associate the views to volumes and other views.

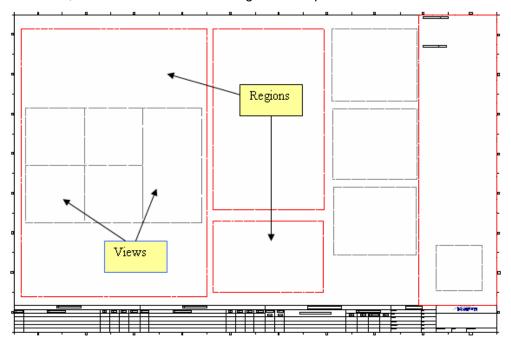
In 3D modeling tasks such as Common, you can use the **Tools > Drawing Console** command to create a composed drawing component and the drawings associated with it. In the Drawings and Reports task, you can create the composed drawing component, but you must use the 3D modeling tasks to associate the drawing to objects in the model.

- 1. In the Drawings and Reports task, use the **Tools > Define Layout Style** command to create layout styles you need for your composed drawing regions, then use the layout styles when you create your layout templates with the **Tools > Edit Layout Template** command.
- 2. Create a folder for the drawing component in the **Management Console**, or go to a 3D task and use **Tools > Drawing Console** to create the component.
- 3. In the 3D modeling task, use the **Drawing Console** to create new composed drawings. Right-click the component, and select **New Drawing**. For more information, see *New Drawing* in the *Orthographic Drawings User's Guide*, available from **Help > Printable Guides**. You can also use **Edit** on the shortcut menu to view or modify existing composed drawings. You can select **Tools > Snapshot View** to add snapshots of the model to composed drawings. For additional information, see the *Common User's Guide* available from **Help > Printable Guides**.
- 4. When the composed drawing is open in the SmartSketch Drawing Editor, you can add views, associate views to volumes or other views, remove view associations, add dependent report views, and modify the layout of the drawing sheet. For more information, see *Create a new composed drawing* (on page 205) in the *Orthographic Drawings User's Guide*. For information on the Drawings Compose toolbar in plant mode, see *Working with 3D Task Drawings* in the *SmartSketch Drawing Editor Help*. For information on the Drawings Compose Toolbar in marine mode, see *Working with Marine 3D Drawings* in the *SmartSketch Drawing Editor Help*.
- 5. After saving the drawing in SmartSketch Drawing Editor, return to the **Drawing Console** to do additional editing, update the drawing, and revise and publish the drawing. After the composed drawing is created, it is just like any other drawing document and can be managed in the Drawings and Reports task. For information on the 2D commands available for editing, see the SmartSketch Drawing Editor Help.

NOTE If the drawing document you are looking at in the **Detail View** has a yellow icon (for example:), the drawing document is a version 6.1 legacy Snapshot drawing. You should use the **Tools > Convert Legacy Snapshots** command to convert this document to a Composed Drawing for use in the current version of the software. If you do not convert the legacy snapshot drawing, you cannot perform edit operations on the drawing, including update, revise, and publish. For more information, see *Convert Legacy Snapshots* (on page 240) in the *Orthographic Drawings User's Guide*.

Drawing Regions and Drawing Views

A region is a container that manages drawing views. The layout style associated with the region dictates the order and placement of the managed drawing views. Drawing views are associated with objects, volumes, or other drawing views. You place regions and views on a composed drawing when it is open in SmartSketch Drawing Editor. For more information on placing regions and views, see the *SmartSketch Drawing Editor Help*.



Regions and drawing views behave as follows:

- You can create a drawing view outside a region, but this makes the drawing view an "unmanaged view," meaning the properties of the region do not impact the drawing view. However, if a view is inside a region or touching a region, the region manages the drawing view. When you update the drawing, the software associates the drawing view with the region and updates it based on the applied region layout style.
- If a region refuses to accept a drawing view (for example, if the region is defined for three views, and you are attempting to add a fourth view), the drawing view is added as an unmanaged view, just outside and to the upper left of the region. If another region occupies this space in the drawing area, the unmanaged view is placed as close to the original region as possible.
- If a drawing view "straddles" two or more regions, the region that contains more of the drawing view manages it. If the drawing view equally straddles two or more regions, the software uses the first drawing view point to measure distance and determine which region manages the drawing view.
- If a region contained a drawing view and the drawing view properties make it ineligible for the region, the software removes the drawing view from the region automatically and places it in the upper left of the drawing area, outside of other regions.

Layout Templates and Layout Styles

Composed drawings use layout templates and layout styles to assist you in creating reusable drawings. The software combines layout templates with drawing borders to create composed drawings. Layouts also use regions as containers to manage drawing views. You also define a layout style associated with the region. The layout style dictates the order and placement of the managed drawing views. For more information on defining layout styles, see Define Layout Style Command (Tools Menu). For additional information on editing layout templates, see Edit Layout Template Command (Tools Menu).

Drawing Borders

When you create a composed drawing in a 3D task, you specify a border template to use with the drawing. After the drawing is created, you can switch the border associated with the drawing using the **Switch Border** command on the shortcut menu of the drawing. For more information, see *Switch Border* (on page 210) in the *Orthographic Drawings User's Guide*.

Composed Drawings Common Tasks

The following tasks are used frequently when you create composed drawings. Some of the tasks are performed within the Drawings and Reports task or other 3D modeling tasks, and others are performed while the drawing is open in **SmartSketch Drawing Editor**.

Create Layout and View Styles, and Edit Templates for Composed Drawings

You use a combination of commands in the Drawings and Reports task to create styles used in your composed drawings and to design the drawing appearance.

- The views you create in a composed drawing require view styles. For more information, see "Define View Style Command (Tools Menu)" in the *Drawings and Reports Reference Data Guide* available from **Help > Printable Guides**.
- Create layout styles to define regions on a layout template used with composed drawings.
 For more information, see "Define Layout Style Command" in the *Drawings and Reports Reference Data Guide* available from Help > Printable Guides.
- Edit border templates to include additional graphics in the border area of a composed drawing. For more information, see "Edit Border Template Command (Tools Menu)" in the Drawings and Reports Reference Data Guide available from Help > Printable Guides.
- Composed drawings combine a layout template with a border template to create drawings that have the same positioning of views but different content. For more information, see "Edit Layout Template Command" in the *Drawings and Reports Reference Data Guide* available from Help > Printable Guides.

Create a Composed Drawing Component

You can create a Composed Drawing component either in the Drawings and Reports task **Management Console**, or by using the **Tools > Drawings Console** command in one of the 3D modeling tasks. For more information, see *Add a package of components* (on page 42), or see *Drawing Console Command* in the *Common User's Guide* available from **Help > Printable Guides**.

Create a Volume for a Composed Drawing

Composed drawings must be associated with a volume in the 3D model before they can be updated successfully. In the Space Management task, you can create volumes to use in your drawings. For more information, see the *Space Management User's Guide* available from **Help** > **Printable Guides**.

Create a Composed Drawing

In a 3D task, create a new composed drawing using the **Tools > Drawing Console** command. Select and right-click the composed drawing component, and select **New Drawing** on the shortcut menu. The software creates the composed drawing based on the layout template and border template you specify. You can also open existing drawings in a 3D task by right-clicking a document in the **Drawing Console** and selecting **Edit**. For more information on creating new composed drawings, see *Create a new composed drawing* (on page 205). To edit an existing composed drawing, see *Edit a Composed Drawing* (on page 404).

Update a Composed Drawing Document

Update the composed drawing document. For more information, see *Updating Documents* (on page 78).

Viewing the Drawing Log

You can view the drawing log to see any messages associated with the drawing. For more information, see *View Log Command* (on page 88).

Set Drawing Properties

You can specify the properties for the drawing component or drawing documents by right-clicking and selecting **Properties**. For more information, see *Edit document properties* (on page 48).

Comparing 2D Drawing Objects to 3D Model Objects

You can open a drawing document and compare the 2D drawing objects in the drawing document to 3D model objects. For more information, see *Compare 2D Drawing Object to 3D Model Object* (on page 362).

Edit the Composed Drawing

In many cases, you need to include additional labels, dimensions, or place new views on the composed drawing after it is created. For more information, see *Edit a Composed Drawing* (on page 404). This is an optional task and is not required to create composed drawings.

Publish the Composed Drawing

Publish the composed drawing documents. You can publish only if your model has been registered using the SmartPlant Registration Wizard. For more information, see *Publishing Documents* (on page 424).

▶ NOTE The viewable files created when you publish drawings and reports provide relationship links to the 3D Model Data. You must also publish the 3D Model Data to provide the navigation between the viewable files and the 3D Model Data. For more information, see 3D Model Data (on page 188).

Convert a Version 6.1 Snapshot Drawing to a Composed Drawing

If a drawing document in the **Detail View** has a yellow icon (for example: (a), the drawing document is a version 6.1 legacy Snapshot drawing. You should use the **Tools > Convert Legacy Snapshots** command to convert this document to a Composed Drawing for use in the current version of the software. If you do not convert the legacy snapshot drawing, you cannot perform edit operations on the drawing, including update, revise, and publish. For more information, see *Convert Legacy Snapshots* (on page 240).

See Also

Drawings and Reports Naming Rules (on page 95)
Layers (SmartSketch Drawing Editor Tools Menu) (on page 402) (Marine mode only)

New Drawing

Creates a new composed drawing. You access this command on the shortcut menu for a composed drawing component in the **Drawing Console** in a 3D modeling task such as Common. When you select **New Drawing**, the **Drawing Sheet General Properties** dialog box displays, which allows you to set the properties for the new composed drawing. When you click **OK**, the drawing opens in SmartSketch Drawing Editor so you can add views, associate views to objects or other views, and modify the drawing area of the composed drawing. For more information on commands available to edit the composed drawing, see the *SmartSketch Drawing Editor Help*.

■ NOTE You must create at least one folder in the **Console** to provide a location for the new composed drawings to be stored. You can also create your composed drawing components in the Drawings and Reports task. For more information on the **Drawing Console**, see the *Common User's Guide*.

What do you want to do?

- Create a new composed drawing (on page 205)
- Create a drawing using WBS objects (on page 208)

Create a New Composed Drawing

■ NOTES

- For information on the Drawings Compose toolbar used in this procedure, see *Working with Drawings and Reports and SmartSketch Drawing Editor* (on page 253).
- For information on the 2D commands available for editing, see the *SmartSketch Drawing Editor Help*.
- Unmanaged drawing views do not move or resize when updating composed drawings or when creating a new composed drawing from a layout template.

Set Up a Drawing

 In the Space Management task, create volumes to use in your drawings. For more information, see the Space Management User's Guide available from Help > Printable Guides. 2. In a 3D modeling task such as Common, select **Tools > Drawing Console**.

The **Drawing Console** opens, showing all of the existing folders and components available for creating drawings.

3. To create a new folder and component, right-click the drawing root in the Management Console, and select **New...**.

The **Add Component** dialog box displays.

- 4. Select Folder on the General tab, and click OK.
- 5. Right-click the folder and select **Rename** to edit the name of the folder.
- 6. Right-click the folder and select **New...**.

The Add Component dialog box displays.

- 7. Select the **Discrete** Composed Drawings component on the General tab.
- 8. Right-click the composed drawings component, and select **Rename** to edit the name of the component.

Create a Drawing Sheet

1. Right-click a composed drawings component in the **Drawing Console** and select **New Drawing**.

The **Drawing Sheet General Properties** dialog box displays. For more information, see Drawing Sheet General Properties Dialog Box (on page 209).

- 2. Select a rule for the Name Rule.
- 3. Type a value for Name, if User Defined is selected for Name Rule.
- 4. Select a layout template by selecting **More** in the **Value** field for the **Layout Template** property.

The **Select Template** dialog box displays. For more information, see Select Template Dialog Box (on page 210).

5. Select a border template by selecting **More** in the **Value** field for the **Border Template** property.

The **Select Template** dialog box displays. For more information, see Select Template Dialog Box (on page 210).

- 6. If you are using batch processing when updating, printing, or refreshing this drawing document, select **Update Settings** for the **Category**, and type a **Batch Processing Timeout** value. The default value is 40 minutes.
- 7. Click **OK** to accept the composed drawing sheet properties. The software "stretches" the layout template you selected to fit the drawing area of the border template you selected and opens the new drawing in **SmartSketch Drawing Editor**.

Create a Drawing View

- 1. Select the SmartSketch Drawing Editor window to make it the active window.
 - TIP The SmartSketch Drawing Editor window may be hidden behind the 3D application window.

- 2. In SmartSketch Drawing Editor, click the Place View Micromand. Click and drag in the drawing area to place a graphic view.

The **Drawing View Properties** dialog box displays. For more information, see Drawing View Properties Dialog Box (Place View Command) - Composed Drawings (on page 276).

3. On the View tab, select More for the Style property.

The Select View Style dialog box displays.

- 4. Select a view style, and click OK.
 - **NOTE** For more information on rule set view style definitions, see *Define View Style Command (Tools Menu)* in the *Drawings and Reports Reference Data Guide*. You can access the guide using the **Help > Printable Guides** command in the Drawings and Reports task.
- 5. On the **View** tab, type a value for **Name**, and select a **Coordinate System**, **Scale Family**, and **User Selected Scale**.
- 6. Click OK.
- 7. Select the graphic view in the drawing area, and click Associate Objects to View 3.
- 8. Select the 3D application window to make it the active window. The 3D application **Associate** ribbon is available. Select a volume to populate the drawing view.
 - **NOTE** You can select a volume from the 3D environment, or from the **Space** tab in the Workspace Explorer.
- 9. Return to the SmartSketch Drawing Editor window to save and update the drawing.

Create a Report View

- Select the SmartSketch Drawing Editor window to make it the active window.
- 2. In SmartSketch Drawing Editor, click the Place Report View Command.
- 3. Click and drag in the drawing area to place a non-graphical view.
 - The Report Properties dialog box displays.
- 4. On the Report View tab, select More for the Report Template property.
 - The Select Template dialog box displays.
- 5. Select a report template, and click **OK**.
 - **NOTE** For more information, see Select Template Dialog Box in the SmartSketch Drawing Editor Help.
 - ★ IMPORTANT For a composed drawing in marine mode, you can only place report views that use dependent view templates. These are templates that are associated with a graphic drawing view. In the delivered reference data, the templates are in folders named "Dependent View Reports."
- 6. Set the output format of the report using the Report Output Format and Report Justification properties. For more information, see *Report Properties Dialog Box (Place Report View Command)* in the *SmartSketch Drawing Editor* Help.
- 7. Type a value for **Name** on the **Report Properties** dialog box, and click **OK**.

- 8. Select the non-graphical report view in the drawing area, and click **Associate Objects to View** ...
- 9. Select the 3D application window to make it the active window. The 3D application **Associate** ribbon is available. Select a graphic **View** to associate with the report view.
 - NOTE The graphic view names from the 2D drawing area are listed in the View field.
- 10. Click **Finish** on the **Associate** ribbon to complete the association of a drawing view to the report view.

Add a Section or Detail View to a Drawing

To add a section or detail view, see *Place a Section View* (on page 355) and *Place a Detail View* (on page 359).

Create Additional Views

- 1. Select the SmartSketch Drawing Editor window to make it the active window.
- 2. Use the commands on the Drawings Compose toolbar to place and associate additional views, place drawing regions, remove view associations if necessary, and perform other editing tasks on the drawing sheet. For information on the Drawings Compose toolbar, see *Working with Drawings and Reports and SmartSketch Drawing Editor* (on page 253).

Save a Drawing after Editing

- 1. Click File > Save.
- 2. Click File > Exit.

The SmartSketch Drawing Editor window closes.

Update the Drawing

- 1. In the 3D application window, select **Tools > Drawing Console**.
- 2. Navigate through the hierarchy to the drawing.
- 3. Right-click the drawing and select **Update Now**.
 - TIP You can also update a single view by right-clicking on a drawing view inside the SmartSketch Drawing Editor and selecting Update View.
- 4. After the drawing is up-to-date, right-click the drawing and select **Edit** to view or edit the drawing in **SmartSketch Drawing Editor**.

■ NOTES

- When the drawing is out-of-date, a red X icon × is superimposed on the drawing icon.
- While the drawing is updating, an hourglass icon \square is superimposed on the drawing icon.
- When the drawing is up-to-date, a green check icon ✓ is superimposed on the drawing icon.

Create a drawing using WBS objects

Work Breakdown Structure (WBS) Projects manage the composition of the plant based on the construction work to be completed. For example, new and existing objects in the model can be

symbolized differently by assigning them to a WBS Project. Drawing view styles can also be configured to symbolize the difference between new and existing projects.

The following procedure shows how to create a drawing that includes WBS Project objects.

Assign Objects to WBS Project

Objects must be included in a WBS Project before they can symbolize correctly in a drawing. For more information on using WBS Project, see *Project Menu* in the *Common User's Guide*.

Create a Parameterized/Asking Filter

A parameterized, or asking, filter must be created before configuring a custom view style for the drawing. For more information on creating a new asking filter, see *Create a new asking filter* in the *Common User's Guide*.

***IMPORTANT** When creating the filter, include the appropriate WBS Project(s) in the **Work Breakdown Structure** tab of the **New Filter Properties** dialog box.

Create a View Style for WBS Objects

You must create a custom view style that includes the parameterized filter created in the previous step. For more information on creating a custom view style, see *Create an Orthographic Drawing View Style* in the *Drawings and Reports Reference Data Guide*.

Select WBS Project on Drawing Properties and Update

1. Right-click on a drawing in the **Drawing Console** or Drawing and Reports task, and select **Properties**.

The **Drawing Properties** dialog box displays.

- 2. Select the Style tab.
- In the WBS Project row, change the Value to the appropriate WBS Project. You can also change the Behavior to Inherit, Override, or Force Override. The default behavior is set to Inherit. For more information on the Style tab, see Style Tab (Properties Dialog Box) (on page 55).
- 4. Click OK.
- 5. Update the drawing(s).

NOTE If a property in the WBS Project field is changed, the entire drawing is marked out-of-date in order to resymbolize the appropriate WBS items when an update is performed.

See Also

Edit a Composed Drawing (on page 404)

Drawing Sheet General Properties Dialog Box

Displays drawing sheet properties for review and editing. This dialog box displays when you right click a drawing component in the **Drawing Console** and select **New Drawing**.

Location

Allows you to select either a composed drawing component or a folder from the Drawings and Reports **Management Console**. If you select a folder, the software automatically creates a new composed drawing component. For more information, see Select Drawings

Component Dialog Box.

Name

Specifies the name of the object. If a **Name Rule** is specified, then the software uses that rule to determine this name. If the **Name Rule** value is **User Defined**, then you must type a name in this box.

Name Rule

Displays the available name rules for the selected object. Specify the naming rule to use to name the object. You can select one of the listed rules, or you can select **User Defined** to specify the name yourself in the **(Name)** box.

Layout Template

Specifies the template for the drawing layout. For more information, see Select Template Dialog Box.

Border Template

Specifies the template for the drawing border. For more information, see Select Template Dialog Box.

NOTE The software uses a combination of the layout template and the border template to create the drawing.

OK

Opens the new composed drawing in **SmartSketch Drawing Editor** so you can place views, associate views to objects, and add other modifications to the drawing area using **SmartSketch Drawing Editor** commands. For more information, see the *SmartSketch Drawing Editor Help*.

See Also

New Drawing (on page 205)

Select Template Dialog Box

Specifies a template for the layout or border of a drawing.

To open this dialog box, open the **Drawing Sheet General Properties** dialog box, and click **More** in the **Value** fields for the **Layout Template** or **Border Template** properties. Select a template from those available in the hierarchy, then click **OK** to return to the **Drawing Sheet General Properties** dialog box.

See Also

New Drawing (on page 205)

Switch Border

Allows you to switch the border template associated with the composed drawing documents selected in the Detail View. This command is available on the shortcut menu when you select a composed drawing document. It displays the **Drawing Sheet Properties** dialog box so you can change the associated border template.

Editing a border template in the SharedContent folder does not affect an existing drawing because the border template associated to a drawing is stored in the database. To change the

border template used by an existing drawing, you must replace the stored border template using the **Switch Border** command.

After switching the border template, the software computes a new position and size for any regions and managed views contained in the drawing. The shift and resize of the view is proportional to the size of the drawing areas in the border templates. If the border template does not contain a drawing area, the drawing boundary is computed automatically.

The software does not distinguish between different types of drawing views (report, key plan, and graphic views) when switching the border.

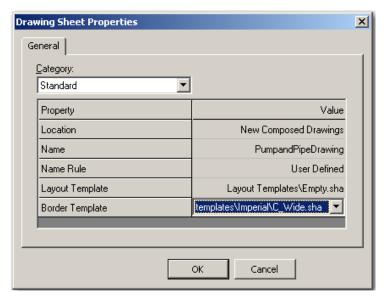
■ NOTES

- If you switch the border template of a drawing, views may resize or reposition with the new border template based on the following conditions:
 - Managed views are proportionately resized and repositioned according to the size of the new border template.
 - Unmanaged views are not resized and repositioned.
- Drawings must be updated after switching the border template in order to ensure all automated annotation is positioned correctly on the drawing.
- For views that are resized after the border template switch, drawing views using Fit to Scale show the same content. Views with a scale may have content clipped out if the view is made smaller after the switch.

For more information, see Place Region Command (on page 296).

Drawing Sheet Properties Dialog Box

Shows the drawing sheet properties associated with the selected drawing document(s) and allows you to change the border template file. When displayed for the **Switch Border** command on selected composed drawing documents, the only property available is **Border Template**. Select **More** in the **Value** field to display the **Select Template** dialog box and specify a new border template. Click **OK** to associate the new border template file to the selected drawing documents.



You need to update the drawings to regenerate them with the new border template file.

See Also

Switch Border (on page 210)

SECTION 14

Orthographic Drawings by Query

You can generate orthographic (volume) drawings using a filter-based query. The Orthographic Drawing by Query component allows you to create drawings for many objects in the model all in the same manner. This component does not require physical volumes in the model. The software uses a filter-based query to collect the objects and document them automatically.

For example, Hanger drawings use parts that can be queried from the database. There are many of them, and in most cases, they are documented the same way. Using the Orthographic Drawing by Query component means you do not have to place hundreds of drawing volumes to document each type of part or assembly.

The basic workflow for creating Orthographic Drawings by Query is as follows:

- Create an Orthographic Drawing by Query component that specifies what you want to document.
- Create a Query Manager that says where to look for the data.
- Run the filter-based query.
- Create the drawings.
- Update the drawings.
- Publish the drawings to a viewable graphic file; no physical data is published.

Administrator Setup

Your administrator sets up appropriate templates to use with the Orthographic Drawings by Query component. The administrator is also responsible for creating filters that define what to look for and filters that specify where in the model to look for the objects. The template and the "what" filter information within the component are saved as a package.

Drawings by Query Manager

The Drawings by Query Manager uses the normal filter to specify where to look for the objects included in the drawing. You do not use an asking filter with the Drawings by Query Manager.

For more information on the filters necessary for setting up a Drawings by Query component, see *Drawings by Query Filters* (on page 216) in the *Orthographic Drawings User's Guide*.

Navigator Rules

The navigator rules for Orthographic Drawings by Query tell the software how to traverse elements to be included in the range for the drawing view objects. The navigator rules can also return separate object collections with ranges that are included in the 3D object range. If no navigator rule is specified for a drawing view, the drawing object collection includes everything in the 3D object range. The delivered rules are as follows

- HngSupSimpleNavigator.dll Returns support components, supporting objects, supported objects, and design children. This rule also returns the control points on the components. This rule is specific to objects created in the Hangers and Support task.
- HngSupRangeNavigator.dll Returns support components, supporting objects, supported objects, and design children collections that extend the Hangers and Support range. This rule also returns the control points on the components. This rule is specific to objects created in the Hangers and Support task and operates similarly to the HngSupSimpleNavigator.dll rule.
- HngSupNoSupportingRange.dll Returns the same components and objects as HngSupRangeNavigator.dll, excluding the supporting objects.
- DrawingSpoolNavigator.dll Returns the spool, its connected parts, and their features.
 This rule is specific to pipe spools.
- AssemblyNavigator.dll Returns the assemblies, pipe spool, penetration spool, its connected parts, and their features. This rule is specific to assemblies.
- SystemRangeNavigator.dll Returns the Equipment objects, as specified by the "what" filter. It also returns design hierarchy children of the Equipment, including any Route objects that are children of the Equipment (Runs, Parts, and Features). This rule is specific to Equipment objects.

When you edit the template associated to an Orthographic Drawing by Query component, such as the **HgrSup3View** package, you can set the navigator rule assigned to each drawing view. Right-click the component or package and select **Edit Template** to open the drawing template in **SmartSketch Drawing Editor**. Select and right-click a drawing view to display the **Drawing View Properties** and set the navigator rule. For more information, see the *SmartSketch Drawing Editor Help*.

Orthographic Drawings by Query Common Tasks

The following tasks are used when you create orthographic drawings by query. For more information on the filters required to create Orthographic Drawings by Query, see *Drawings by Query Filters* (on page 216).

Setup an Orthographic Drawing by Query Component

Setup allows you to select the appropriate filter for an Orthographic Drawing by Query component. When you select a filter in **Setup**, you are specifying the "what" portion of the query. In other words, you are specifying the objects to be included in the drawing. You can also specify how much extra space you want to include around the objects. To setup an Orthographic Drawing by Query component with the appropriate filter, right-click the component and select **Setup**. For more information, see *Setup an Orthographic Drawing by Query component* (on page 218).

Edit the Template

Edit the template associated to an Orthographic Drawing by Query component. Editing the template allows you to set the navigation rule assigned to each drawing view. Right-click the component or package, and select **Edit Template** to open the drawing template in **SmartSketch Drawing Editor**. Select and right-click a drawing view to display the **Drawing View Properties**, and set the navigator rule. For more information, see the *SmartSketch Drawing Editor Help*.

You can also edit the drawing border template to meet your output requirements. For more information, see "Edit Border Template Command (Tools Menu) in the *Drawings and Reports Reference Data Guide*.

Save the Orthographic Drawing Component as a Package

Save the Orthographic Drawing by Query component as a package so you can join it with a **Drawings by Query Manager** to generate the query drawings. For more information, see *Save a package* (on page 77).

Add a Drawings by Query Manager

Create the **Drawings by Query Manager**, then set it up to specify the "where" portion of the query for the Orthographic and Piping Isometric Drawing components. For more information, see *Set up a Drawings by Query Manager component* (on page 219).

Run the Query

Execute the query specified by the Orthographic Drawing by Query component and the **Drawings by Query Manager**. For more information, see *Run Query (Shortcut Menu)* (on page 70).

Create or Update the Drawings

Create or update drawings by right-clicking the component and selecting the appropriate command. For more information, see *Updating Documents* (on page 78).

Viewing the Drawing Log

Display the drawing log, which details any messages associated with the drawing. For more information, see *View Log Command* (on page 88).

Set Drawing Properties

Specify the properties for the drawing component or drawing documents by right-clicking and selecting **Properties**. For more information, see *Edit document properties* (on page 48).

Comparing 2D Drawing Objects to 3D Model Objects

Open or edit a drawing document and compare the 2D drawing objects in the drawing document to 3D model objects. For more information, see *Compare 2D Drawing Object to 3D Model Object* (on page 362).

Publish the Orthographic Drawing

Publish the Orthographic Drawing by Query documents. You can publish documents only if your model has been registered using the SmartPlant Registration Wizard. For more information, see *Publishing Documents* (on page 424).

■ NOTE The viewable files created when you publish drawings and reports provide relationship links to the 3D Model Data. You must also publish the 3D Model Data to provide the navigation between the viewable files and the 3D Model Data. For more information, see 3D Model Data (on page 188).

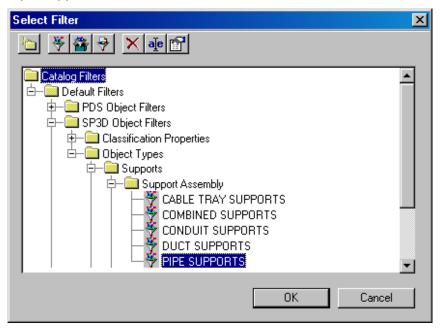
See Also

Layers (SmartSketch Drawing Editor Tools Menu) (on page 402) Drawings and Reports Naming Rules (on page 95)

Drawings by Query Filters

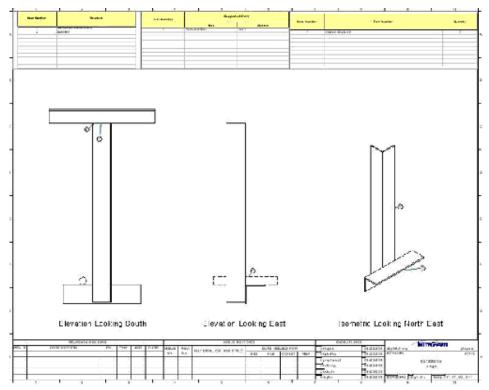
Filters determine the contents of the documents produced by Drawings by Query components (3D Model By Query, Orthographic, Isogen Isometric, and Drawings by Query Manager). For Orthographic and Isogen Isometric Drawings by Query components, the **Setup** command specifies a *what* filter that access the model database and determines which objects are included in the drawings, as well as which hierarchy is traversed to create a tree in the **Management Console**. **Setup** then defines a Drawings by Query Manager component to specify a *where* filter that determines the position in the hierarchy and, therefore, the location of the objects included in the drawing.

For example, if you are creating Orthographic drawings that include pipe supports, you create a new Orthographic Drawings by Query component, and then run **Setup** on the component to select a normal filter that returns only pipe support objects. For example, you might choose the **Pipe Supports** filter as shown below:



After saving your Orthographic Drawings by Query component setup as a package, you then define a Drawings by Query Manager component to determine *where* in the model to collect the objects you specified in the *what* filter. When you run **Setup** on the Drawings by Query Manager component, you select a normal filter.

When you run the query, the software compounds the *where* filter with the *what* filter to return the needed objects. Each object is documented in a drawing using the template and rules that you setup for the package. The following graphic shows an example drawing using the **Pipe Supports** filter:



For more information on defining filters, see the *Common User's Guide* available from **Help > Printable Guides**.

- **NOTE** A *what* filter can specify non-graphical objects, such as various system nodes in the **System** tab, folders in the **Space** tab, WBS projects and items in the **WBS** tab, and Reference 3D Model nodes in the **Reference 3D** tab of the **Workspace Explorer**. When you specify non-graphical objects in the *what* filter, you must also specify a Navigator Rule in the **Drawing View Properties** dialog box.
- TIP You can use the **SystemRangeNavigator.dll** Navigator Rule to include all graphical objects under a non-graphical object. For more information on navigation rules, see the *Navigator Rules* section in *Orthographic Drawings by Query* (on page 213).

See Also

Orthographic Drawings by Query Common Tasks (on page 214)

Setup (Orthographic Drawing by Query Component)

Sets component options for creating Orthographic Drawings by Query. This command is available when you right-click the Orthographic Drawing by Query component.

Setup Dialog Box (Orthographic Drawing by Query Component) (on page 218)

Setup an Orthographic Drawing by Query component

1. Right-click the folder where you want to store the new Orthographic Drawing by Query component, and select **New > Orthographic Drawing by Query**.

The Orthographic Drawing by Query component is created in the folder.

2. Right-click the Orthographic Drawing by Query component and select Setup.

The Select Filter dialog box displays.

- 3. Select the filter that specifies the objects to include in the drawing. Click **Properties** to display the current filter properties. For more information on filters for Drawings by Query, see *Drawings* by *Query Filters* (on page 216).
- 4. If you want extra space included around the objects, specify a **Volume Growth** value. The larger the value, the more space is included around each object.
- 5. If you want to include the supported objects in the volume, set **Expand Volume to Include Supported Objects** to **On**.

■ NOTES

- This option is only available for Hangers and Supports drawings that use the HngSupRangeNavigator.dll, HngSupNoSupportingRangeNav.dll, or SystemRangeNavigator.dll navigator rules.
- You can control this option by editing the XML file for the template. To turn Expand Volume to Include Supported Objects on, set the <cli>clipsupported> node to -1. To turn it off, set the node to 0.

6. Click OK.

To generate orthographic drawings with using the component, you need to save it as a package and associate it to a **Drawings by Query Manager**. For more information, see *Set up a Drawings by Query Manager component* (on page 219).

■ NOTES

- To delete a component, right-click it and select Delete.
- To rename a component, right-click it and select Rename.

Setup Dialog Box (Orthographic Drawing by Query Component)

Filter

Identifies the filter that defines the *what* portion of the query. The filter determines the objects included in the drawings when they are generated. Select **More** in the **Filter** list to display the **Select Filter** dialog box. Click **Properties** to display the **Filter Properties** dialog box. For more information on filters for Orthographic Drawings by Query, see *Drawings by Query Filters* (on page 216).

Volume Growth

Specifies the *growth range*, or extra space, that you want to include around the objects identified by the query. The larger the setting, the more space is included around the object.

Expand Volume to Include Supported Objects

Expands the drawing volume to fit the supported object. If the supported object is not a straight feature, the entire graphic of the supported object is shown. If the supported object is a straight feature, the segment encapsulated in the drawing volume is shown.

■ NOTES

- This option is only available for Hangers and Supports drawings that use the HngSupRangeNavigator.dll, HngSupNoSupportingRangeNav.dll, or SystemRangeNavigator.dll navigator rules.
- You can control this option by editing the xml file for the template. To turn Expand Volume to Include Supported Objects on, set the <cli>clipsupported node to -1. To turn it off, set the node to 0.

See Also

Setup (Orthographic Drawing by Query Component) (on page 217)

Select Filter

Specifies a filter for orthographic or Isogen isometric drawings created by Drawings by Query components. The filter narrows the objects returned for the drawings.

For more information on setting up filters for Drawings by Query components, see *Drawings by Query Filters* (on page 216).

Select Filter Dialog Box (on page 122)

Setup (Drawings by Query Manager Component Shortcut Menu)

Sets options for creating a Drawings by Query Manager component. This command is available on the shortcut menu when you right-click a Drawings by Query Manager component.

The Drawings by Query Manager component is used in conjunction with other components, such as the Orthographic and Isogen Isometric Drawing by Query components, to complete the query for objects in the model. The Drawings by Query Manager provides the filter that specifies the *where* side of the query. It tells the query *where* to look for the objects specified by the component *what* filter.

Setup Dialog Box (Drawings by Query Manager Component) (on page 220)

Set up a Drawings by Query Manager component

The Drawings by Query Manager component works in conjunction with the Orthographic Drawing by Query and Isogen Isometric Drawing by Query components. Before using this command, you must create packages for your Orthographic Drawing by Query and Isogen Isometric Drawing by Query components. For more information, see *Save Package Command* (on page 76).

1. Right-click the folder in which to create your Drawings by Query Manager.

The software displays the **Add Component** dialog box.

- TIP You can store the Drawings by Query Manager anywhere in the **Console**, but it is best to store it in the same location as the components with which it works.
- 2. Select the Drawings by Query Manager component, and then click **OK**.
 - The software creates the Drawings by Query Manager component in the folder.
- 3. Right-click the Drawings by Query Manager component, and then select **Setup** to specify the properties for the component.
 - The software displays **Setup** dialog box.
- 4. Specify a filter in the Filter field. The list shows the most recently selected filters. Select More in the list to display the Select Filter dialog box, and then specify a filter. Click Properties to display the current filter properties.
 - **NOTE** The filter that you select is the *where* portion of the query, as opposed to the *what* portion specified when you set up the Orthographic Drawing by Query or Isogen Isometric Drawing by Query component. The filter that you specify here tells where in the model you want to look for the objects. For more information on filters for Drawings by Query, see *Drawings by Query Filters* (on page 216).
- 5. In the **Package** field, specify the Orthographic Drawing by Query or Isogen Isometric Drawing by Query package that you created. The list contains the most recently selected packages. Select **More** to display the **Select Package** dialog box. For example, if you are defining a Drawings by Query Manager for an Orthographic Drawing by Query, select an Orthographic Drawing by Query package.
- 6. Click **OK** to save the settings.

To create the drawings, you need to run the query. For more information, see *Run Query (Shortcut Menu)* (on page 70).

■ NOTES

- To delete a component, right-click the component, and then select **Delete**.
- To rename a component, right-click the component, and then select **Rename**.

Setup Dialog Box (Drawings by Query Manager Component)

Sets options for creating Drawings by Query Manager components.

■ NOTE You must create an Orthographic Drawing by Query, Isogen Isometric Drawing by Query, or 3D Model by Query package before setting up the Drawings by Query Manager component. For more information, see Save Package Command (on page 76).

Filter

Identifies the filter to use to define the *where* portion of the query. The software uses the filter to determine where to look for the objects requested in Orthographic Drawing by Query and Isogen Isometric Drawing by Query components when they are generated. Select **More** in the list to display the **Select Filter** dialog box. Click **Properties** to display the current filter properties. For more information on filters for the Drawings by Query Manager, see *Drawings by Query Filters* (on page 216).

Package

Specifies the package to use in completing the query.

See Also

Setup (Drawings by Query Manager Component Shortcut Menu) (on page 219)

Run Query (Shortcut Menu)

Runs the query associated with the selected Drawings by Query Manager component. For example, if you execute **Run Query** on an Isogen Isometric Drawing by Query package associated with a Drawings by Query Manager, the software looks for piping in the model. The query results display beneath the style in the **Management Console**. You can create isometric drawings from the query results by right-clicking on the component and selecting **Create Drawing(s)**.

If you have an Orthographic Drawing by Query package associated to a Drawings by Query Manager, the **Run Query** command runs the query associated with the components, collects the objects from the database, and builds the information that will be included in the drawing. You can then create orthographic drawings from the query results by running the **Create Drawing(s)** command on the Orthographic Drawing component.

The Run Query command uses the filters specified when you performed Setup components.

■ NOTES

- When working in a Global Workshare Configuration with users logging into both Host and Satellite systems for more than one site, you can encounter problems with the filters defined for a Drawing by Query package. For example, if you create a Filter Root Folder for a particular site, and then you try to run queries for drawings associated to this filter from another site, the Run Query command is unable to update the drawings due to a lack of permission against the Filter Root Folder. You have to move or transfer the Filter Root Folder to the appropriate site where the Run Query command is executed.
- If the selected package was created and added manually, the folder name for the package must match the value for **pkgid** in the XML file. Edit the name of the folder to match the **pkgid** value.
- ★ IMPORTANT In marine mode, this command displays in the Ship Root and Folder shortcut menus for drawings by rule, but is not used with drawing by rule components.

See Also

Orthographic Drawings by Query (on page 213) Create Drawing(s) Command (on page 39)

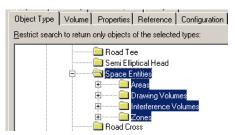
Create Orthographic Drawings by Query for volumes

The following procedure provides an example workflow for creating an Orthographic Drawing by Query to find volumes in the model. The software collects the volumes based on the filters specified and generates a drawing for each volume. The view style associated with the template you are using hides the volume and show the objects inside the volume.

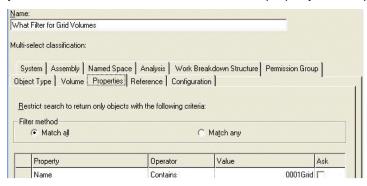
NOTE This method of creating volume drawings is very similar to creating volume drawings using the Volume component in previous versions of the software. You use a template to specify how each volume is managed and placed. The advantage of using an Orthographic Drawing by Query component is that you do not have to place the volumes in the Space Management task as Drawing Volumes. You can use any type of Space Management volume to create the Orthographic Drawings by Query.

This procedure involves 3D tasks as well as the Drawings and Reports task.

- In the Space Management task, create volumes that contain the objects you want to include in your drawings. For example, you can use the Place Volumes By Grid command to create volumes based on your grid system definition. For information on the Place Volume commands, see the Space Management User's Guide available from Help > Printable Guides.
- 2. In any of the 3D tasks, use the **Tools > Select by Filter** command to create a *what* filter to look for the volume objects. For example, on the **Select Filter** dialog box, create a new model filter called **What Filter for Grid Volumes**. Select **Object Type > Space Entities**.



Select the **Properties** tab to define volume properties to simplify the query. For example, you could set the filter to look for the **Name** property with a specific value such as **0001Grid**.



- 3. Go to the Drawings and Reports task and create an Orthographic Drawing by Query component. For more information, see *Setup an Orthographic Drawing by Query component* (on page 218). When you specify the filter during setup of the component, select the *what* filter you create that looks for the volumes.
- 4. Right-click the component and select **Edit Template** to place drawing views for your volumes.
 - NOTE When you edit the template associated to an Orthographic Drawing by Query component, you can set the navigation rule assigned to each drawing view. The navigator rules for Orthographic Drawings by Query tell the software how to traverse elements to be included in the range for the drawing view objects. Right-click the component or package and select Edit Template to open the drawing template in SmartSketch Drawing Editor. Select and right-click a drawing view to display the Drawing View Properties and set the navigator rule. For more information, see the SmartSketch Drawing Editor Help.
- 5. Right-click the component and select **Save Package**. For more information, see *Save Package Command* (on page 76).
- 6. Create a Query Manager to define the *where* side of the query, which determines where in the model to look for the volumes you created.
 - Set up a Drawings by Query Manager component (on page 219)

- Right-click the Drawings by Query Manager, and select Run Query.
- 8. Right-click the **Drawings by Query Manager**, and select **Create Drawing(s)** to generate the drawings based on the objects returned by the query and the Orthographic Drawing By Query package you specified. You can also navigate through the **Management Console** hierarchy and use the **Create Drawing(s)** command from the folder levels.
- 9. Update the drawing documents to save the changes. For more information on updating documents, see *Updating Documents* (on page 78).

See Also

Orthographic Drawings by Query Common Tasks (on page 214)
Setup (Drawings by Query Manager Component Shortcut Menu) (on page 219)
Setup (Orthographic Drawing by Query Component) (on page 217)

Drawing View Properties Dialog Box (Drawing by Query)

Specifies properties for the selected drawing view using drawing by query. You can access this dialog box when you select and right-click on a drawing view and then select **Properties** on the shortcut menu.

Info Tab (Drawing View Properties Dialog Box) (on page 276)

Format Tab (Drawing View Properties Dialog Box) (on page 277)

View Tab (Drawing View Properties Dialog Box) (on page 223)

See Also

Orthographic Drawings by Query (on page 213)

View Tab (Drawing View Properties Dialog Box)

Name

Specifies a name for the view. You must type a name in order to create a view.

Description

Provides a description for the drawing view.

Style

Specifies a view style, which includes rules for filters, updates, and graphics. The view style controls the output characteristics of the view on the generated drawing. The list displays the 10 most recently used view styles in the session. Click **More...** to display the **Select View Style** dialog box.

Use object coordinate system

Select this option to use the object coordinate system.

Orientation

Specifies the orientation of the drawing view.

Scale

Provides the scale of the drawing view.

Navigation Rule

Specifies the type of navigation rule.

Convert report output to text boxes (no Excel)

Select this option to convert the report output to text boxes and not into MS Excel.

Report Justification

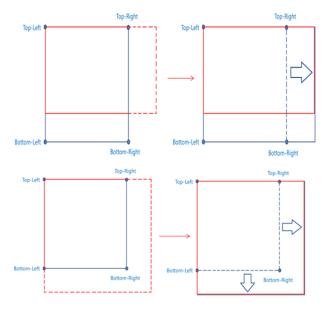
Specifies the justification of the report window. Select **Top-Left**, **Top-Right**, **Center-Center**, **Bottom-Left**, or **Bottom-Right** to align the report to one of these positions. For example, if you select **Top-Right**, the top-right corner of the report window is aligned to the top-right corner of the view.

■ NOTES

- The Report Justification option is enabled when you select Convert report output to text boxes (no Excel) option.
- The report justification properties does not work for dependent reports.
- The Center-Center option is the only justification option that scales the report window to fit the view.
- If you select the Top-Left option, the resulting report view is as shown below.



• If a report view is smaller or larger than a SmartFrame, the SmartFrame will be resized based on the report contents as shown below.



SECTION 15

Volume Drawings

The Volume Drawing component uses a template to create drawings. You can place a view on the template and associate the view with a view style to control the output. To define the contents of the view, you create a drawing volume in the model. Before you create these types of drawings, an administrator should create or edit drawing view styles and templates for use with the drawings. For example, you can place title block labels and reports on the templates to reflect your project or company standards.

After you have your view styles and templates in place, you are ready to start creating drawings. The volume drawings require that the changes occur within the drawing template.

The software allows you to place several views on a template, each with independent scale. In the Space Management task, you can place a volume, which is associated to single or multiple views. The volume you create in Space Management is associated with all the views you create within a volume drawing template. Therefore, in a multiple view volume drawing, you can have multiple views of the same volume, and each view specifies the style, scale, and orientation for the volume associated with the view. If the volume you create is too big for a particular view at its scale, the volume is clipped.

For example, if you place two views in a drawing template with the following property settings:

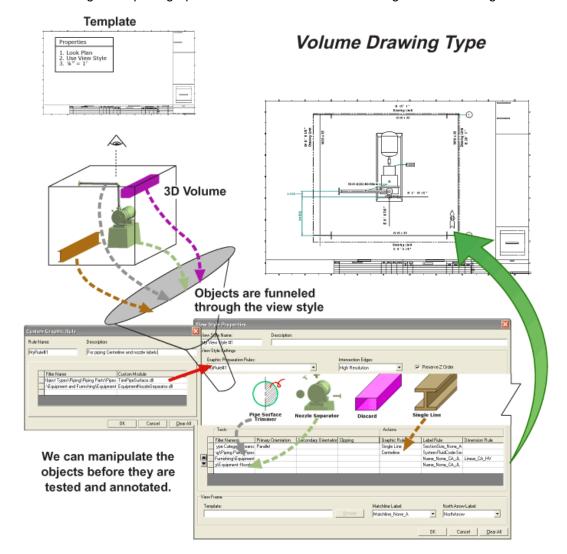
- **DrawingView1** Size 6" x 6", Scale 1" = 1', and Orientation = Plan
- DrawingView2 Size 6" x 6", Scale 1" = 1', and Orientation = North,

you use the **Place Volume By 2-Points** command in the Space Management task from coordinates **(0,0,0)** to **(6,6,6)**. In this case, the volume is exactly the right size based on scale and orientation. The generated drawing contains the complete contents of the volume.

However, if your volume was placed from **(0,0,0)** to **(8,6,6)**, the volume is *too big* for the drawing view on the template. The maximum volume size the view can accommodate is no more than 6 inches on each side. The software centers the volume in the drawing view, effectively giving you the information from **(1,0,0)** to **<7,6,6** in the output drawing.

The overall processing for volume drawings is as follows:

- Identify a drawing volume. This is a query of all the objects that have been created in the database which reside inside the volume borders. The objects do not have to be visible or even in the current workspace to be included.
- When the drawing is created, the software checks the objects from the volume query against the list of filters in the specified view style.
- If an object in the volume matches one of the specified view style filters, the software applies the rules set for the object filter. If the object does not match a filter, it is not included in the drawing view.



The following conceptual graphic shows the workflow for creating volume drawings:

The software applies the view styles in a top-down manner. If an object finds its match in several filters, the last one applied (the bottommost style) overrides any preceding style rules.

When a drawing is updated, it can be published by right-clicking the document and selecting **Publish** on the right-click menu. The document is published as a viewable graphic file; no physical data is published. For more information, see *Publishing Documents* in any of the Smart 3D Drawings user's guides, available from **Help > Printable Guides**.

Volume Drawings Common Tasks

The following tasks are used frequently when you create volume drawings.

Create a Volume Component

Create a Volume component using the **New** command. For more information, see *Add a package of components* (on page 42).

Edit the Volume Component Template

The template for the volume drawing defines the size and location of the frames that will contain the drawing volumes. For more information, see *Edit a volume drawing template* (on page 231). If your volume drawing requires a key plan, you can include it by placing one while in **SmartSketch Drawing Editor**. For more information, see *Place a Key Plan* (on page 390). For additional information, see the *SmartSketch Drawing Editor Help*.

Place Volumes in Space Management

In the Space Management task, create volumes using one of the **Place Drawing Volume** commands on the vertical toolbar. For more information, see *Defining Drawing Volumes* (on page 228). For detailed information on these commands, refer to the *Space Management User's Guide*.

Create the Volume Drawing Document

Right-click the component and select **Create Drawing** to create the volume drawing document. The drawing documents are created. For more information, see *Create Drawing (Volume Drawings)* (on page 229).

Updating Volume Documents

Update the volume drawing document. For more information, see *Updating Documents* (on page 78).

Viewing the Drawing Log

View the drawing log to see any messages associated with the drawing. For more information, see *View Log Command* (on page 88).

Set Drawing Properties

Specify the properties for the drawing component or drawing documents by right-clicking and selecting **Properties**. For more information, see *Edit document properties* (on page 48).

Comparing 2D Drawing Objects to 3D Model Objects

Open a drawing document in a 3D task and compare the 2D drawing objects in the drawing document to 3D model objects. For more information, see *Compare 2D Drawing Object to 3D Model Object* (on page 362).

Manually Add Labels

In many cases, you need to include additional labels or dimensions on the volume drawing after it is created. For more information, see *Place a Manual Label* (on page 325). This is an optional task and is not required to create volume drawings.

Publish the Volume Drawing

Publish the Volume Drawing. You can publish only if your model has been registered using the SmartPlant Registration Wizard. For more information, see *Publishing Documents* (on page 424).

NOTE The viewable files created when you publish drawings and reports provide relationship links to the 3D Model Data. You must also publish the 3D Model Data to provide the navigation between the viewable files and the 3D Model Data. For more information, see 3D Model Data (on page 188).

See Also

Layers (SmartSketch Drawing Editor Tools Menu) (on page 402) Drawings and Reports Naming Rules (on page 95)

Defining Drawing Volumes

Drawing volumes are a critical piece of the volume drawing generation process. Drawing volumes are different from the other drawing components in that they exist in the three-dimensional model as objects.

After you place a drawing view on a volume template, you must define one or more volumes for the view. Volumes clip the results of a query for a drawing view, which displays a two-dimensional representation of the model.

You define a drawing volume with the **Place Drawing Volume** commands in the Space Management task. It is possible to size the volume based on the grid coordinates in the model. Volumes can be placed by view (if single view), by 2-points, by 4-points, and by object selection.

NOTE When you use drawing volumes with key plans, you can specify that the key plan is created with a coordinate system relative to the drawing volume view. For more information on placing key plans, see *Place a Key Plan* (on page 390). For additional information, see the *SmartSketch Drawing Editor Help*.

Volumes are stored in the Model database. When a 3D task such as Piping is active, you can see the defined volumes on the **Space** tab of the **Workspace Explorer**. With this solution, you can locate and manipulate drawing volumes quickly and easily. When you create a drawing volume, the software automatically names the volume and places it in the space hierarchy.

★ IMPORTANT If you delete a drawing volume in the model, the software deletes any associated drawings and views.

See Also

Create a volume drawing (on page 229)

Create Drawing (Volume Drawings)

Generates a drawing for a volume component. Before you can use this command to generate drawings, you must complete the steps below.

- Create at least one Volume Drawing Type component in the hierarchy.
- Define an Orthographic view style using the Tools > Define View Style command in this task
- Create at least one volume drawing view with the Place Drawing Volume commands in the Space Management task.

▶ NOTE If the drawing view you created on the drawing component template is larger than the volume you created using the Place Drawing Volume command, the software centers the volume on the drawing view. If the volume is smaller than the drawing view, the software clips the volume to fit the drawing view according to its scale.

Create a volume drawing

1. In the **Console**, verify that at least one volume drawings component exists. If none exists, add a component for volume drawings.

Add a package of components (on page 42)

Right-click the volume drawings component, and select Edit Template on the shortcut menu.

The Select Template dialog box displays.

- 3. Select a template file.
 - TIP You can modify the templates in this task by clicking Tools > Edit Border Template. For example, place drawing property labels or reports in the title block and save.
- 4. Click **Place Drawing View** to place one or more drawing views on the template.

Place a Drawing View for Volume Drawings (on page 381)

You cannot undo a drawing view delete operation. A message box displays when you press **Delete** for a selected view, providing a chance to cancel the operation.

- 5. Select a view style for the drawing view.
 - TIP You can create and edit view styles by clicking **Tools > Define View Style** in the Drawings and Reports task.
- 6. Add symbols to the drawing view. You may need to click **Symbol Explorer** to display the symbol browser.
 - For example, you can place a north arrow symbol. When the drawing is generated, the north arrow will orient itself according to the view direction.
- 7. Exit and save the template.
- 8. Create one or more drawing volumes to associate with the drawing view by switching to the Space Management task and clicking one of the **Place Drawing Volume** commands.

9. Update the drawing component documents.

■ NOTES

- The software allows you to place several views on a template. In the Space Management task, you can place a volume, which is associated to all the views.
- After you generate drawings, the software does not allow you to edit the view scale. However, you can change the view scale on the template.
- You can export drawings to files by right-clicking a folder, component, or single drawing, and then selecting Save As on the shortcut menu.
- If you are using Office 2003, in Microsoft Excel under Tools > Macro > Security > Trusted
 Publishers tab, check the Trust Access to Visual Basic Project option.
- If you are using Office 2007 or Office 2010, click Microsoft Office to access Excel Option.
 Go to the Trust Center category and select Trust Center Settings. Select the Macro Settings category and check Trust access to the VBA project object model.
- For more information about Microsoft Office and service packs, refer to the *Microsoft web site* (http://www.microsoft.com/) (http://www.microsoft.com/).
- If the drawing view you created on the drawing component template is larger than the volume you created using the Place Drawing Volume command, the software centers the volume on the drawing view. If the volume is smaller than the drawing view, the software clips the volume to fit the drawing view according to its scale.

Edit Template (Volume Drawings Component)

Allows you to modify a template for a volume drawing component in **SmartSketch Drawing Editor**. Use the **Template Toolbar** commands to add a drawing view, report, or key plan.



When you place objects like drawing views, key plan views, report views, and drawing property labels, the software automatically places them on the **DwgTemplate** layer when you save the drawing document. You should not place manual markups on the **DwgTemplate** layer.

If you use other native **SmartSketch Drawing Editor** commands (such as **Place Line** or **Place Dimension**) to add manual markups to the template, put them on the **Default** layer or on a customized layer (for example, a layer named **AnnotationLayer**). This preserves the changes when you update drawings. For more information on layers, see *Layers* (*SmartSketch Drawing Editor Tools Menu*) (on page 402).



Before you click the **Edit Template** command for a volume drawing, you can use the **Tools** > **Edit Border Template** command to place drawing property labels in the title block of a template. For more information, see *Place a Drawing Property Label on a Template* (on page 393).

Edit a volume drawing template

- 1. Right-click a volume drawing component in the **Management Console**, and select **Edit Template**.
- 2. On the **Select Template** dialog box, select a border template for the drawing, and click **OK**. The template opens in **SmartSketch Drawing Editor**.
- 3. In the SmartSketch Drawing Editor window, click Place Drawing View . Click and drag to place a view. You can place multiple volume drawing views.
 - You cannot undo a drawing view delete operation. A message box displays when you press **Delete** for a selected view, providing a chance to cancel the operation.
- 4. On the **Properties** dialog box for each drawing view, provide a name and description for the drawing view.
- 5. Specify a style and scale on the drawing sheet, and set the view orientation for each drawing view.
 - If your volume drawing requires a key plan, you can place one with the **Place Key Plan** command. For more information, see *Place a Key Plan* (on page 390), or the *SmartSketch Drawing Editor Help*.
- 6. Save the changes you made and close **SmartSketch Drawing Editor**.

Drawing View Properties Dialog Box (Volume Drawings)

Specifies properties for the selected drawing view in a volume drawing. You can access this dialog box when you select and right-click on a drawing view and then select **Properties** on the shortcut menu.

Info Tab (Drawing View Properties Dialog Box) (on page 276)

Format Tab (Drawing View Properties Dialog Box) (on page 277)

View Tab (Drawing View Properties Dialog Box) (on page 223)

See Also

Volume Drawings (on page 225)

SECTION 16

Search Folders

Search Folder allows you to search for documents based on common properties such as out-of-date status, approval, or documents that have been published to a certain contract in integrated environment. You create a search folder by right-clicking the root model or a folder in the **Management Console** or the **Drawing Console** and selecting **New** > **Search Folder**.

After running the query defined for a search folder, you can perform such tasks as **Update** or **Publish** as if you are working from the actual component for the documents.

The documents found by a search folder query are listed in the **Detail View**. You are able to interact with the search folder documents just as if you were dealing with the actual components that own the documents. Search folders can also be used for reports.

Search Folder Shortcut Menu

Right-click a search folder component property to display the shortcut menu.

Run Query

Runs the query specified by the search folder setup definition. If you have not run the search folder **Setup** command, this command is not available.

Publish

Publishes a single document or all documents in the imported folder. This command is only available if the model is registered with SmartPlant Foundation.

Update and Publish

Updates and immediately publishes a single document or all documents in the search folder. This command is only available if the model is registered with SmartPlant Foundation.

Setup

Specifies the query to run for the search folder. For more information, see Setup (Search Folder) (on page 234).

NOTE Setup for the search folder does not perform the individual setups for any of the drawing by query documents found in the search folder query and shown in the detail view.

Copy

Copies the search folder. It does not copy the associated documents shown in the detail view area. The search folder setup information is saved with the copy. After you paste the copy to a different location in the **Console** hierarchy, you can run **Setup** again as needed for the new search folder.

Delete

Deletes the search folder. It does not delete any of the documents found in the associated detail view.

Rename

Renames the search folder. It does not affect any of the documents found in the associated detail view.

Revise

Allows you to revise all documents in the search folder without publishing them. This command is only available if the model is registered with SmartPlant Foundation. For more information, see *Revise* (on page 421).

Save Package

Allows you to save the search folder and its definition as a package to be reused in other folder locations in the hierarchy. If you have not run the search folder **Setup** command, this command is not available.

Properties

Displays the **Configuration Properties** for the document.

NOTE To change properties on the document, go to the root location of the document. The search folder does not participate in any propagation of properties to its documents.

■ NOTES

- After a search folder is placed in the console, it follows the same localized naming convention as a folder, such as "New Search Folder" and "New Search Folder (2)."
- Unlike a folder , you cannot create child components in a search folder . This includes pasting existing components; however, you can copy the search folder and paste it elsewhere in the hierarchy.
- Performing Publish, Update and Publish, and Revise on a folder does not include documents under a child search folder. The commands need to be performed separately on the child search folder in order for its documents to be affected.
- You can run commands from the search folder level to modify all of the documents within the folder, or you can modify the individual documents by selecting them in the **Detail View**.

Create a search folder

 Right-click the model root or a folder in the Management Console, and select New > Search Folder.

The search folder is added to the hierarchy.

2. Right-click on the new search folder, and select Setup.

The **Setup** dialog box displays.

- 3. Select More in the Filter list, and select a filter. Click Properties at to display the Filter Properties dialog box.
- Select More in the Start From list, and select a component from the Select Drawings Component dialog box.
- 5. Click OK.
- 6. Right-click the search folder, and select **Run Query**.

The documents returned by the query are added to the Search Folder listing in the Detail

View.

7. To rename the folder, right-click the folder, and select **Rename**, or select the folder, and press **F2** on the keyboard. Type a new name.

Setup (Search Folder)

Sets options for creating a search folder. This command is available on the **Search Folder** shortcut menu and displays the **Setup** dialog box.

NOTE Setup for the search folder does not perform the individual setups for any of the drawing by query documents found in the search folder query and shown in the detail view.

Setup Dialog Box (Search Folder) (on page 234)

Setup Dialog Box (Search Folder)

Specifies a filter that identifies the objects to be included in the search folder query. The **Setup** definition tells the query "where" to look for the objects specified by the component "what" filter.

Filter

Identifies the filter that defines the *what* portion of the query. The software uses the filter to determine the objects included in the drawings when they are generated. Select **More** in the **Filter** list to display the **Select Filter** dialog box. Click **Properties** dialog box. For more information, see *Search Folder Filters* (on page 234).

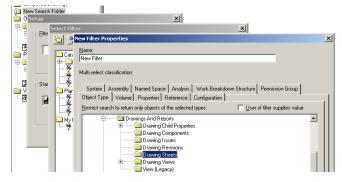
Start From

Specifies the location in the model from which to start the object search. This is the "where" side of the query. Select **More** in the **Start From** combo box to display the **Select Location** dialog box.

Search Folder Filters

Search folders use filters to specify how the component identifies the documents to include in the search folder. You can use Drawings and Reports object properties to define filters when creating search folder components. This allows you to search for documents based on common properties such as out-of-date status, approval, or documents that have been published to a certain contract in integrated environment.

When you run **Setup** on a search folder, you can create filters that check for specific drawing or report object properties.



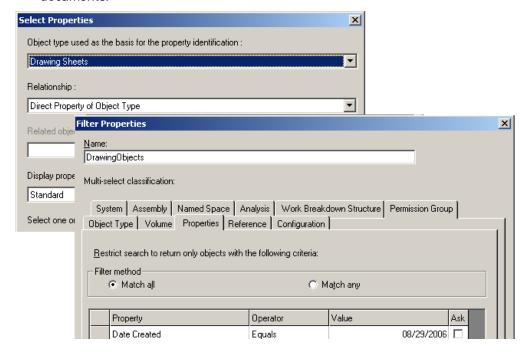
The following examples show how you might create filters to search for specific drawing object properties:

Basic Drawing Document Properties - Title, Area, and Signature

You can use the **Drawing Sheet** object and its properties to look for properties associated with the drawing documents. The following procedure shows how to access the Title, Area, and Signature properties on the drawing and report documents.

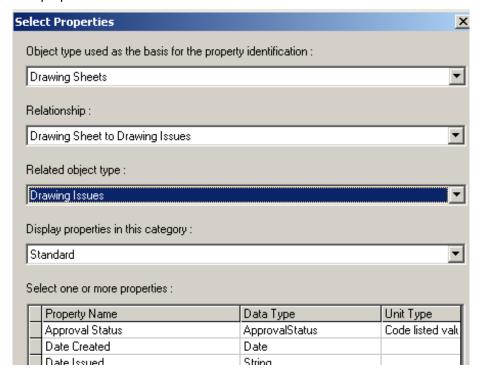
- On the Properties tab of the Filter Properties dialog box, select More in the Property field.
 The Select Properties dialog box displays.
- 2. Under Object type used as the basis for the property identification, select Drawing Sheets.
- 3. Under Relationship, select Direct Properties of Object Type.

You can now search the documents for specified properties under **Select one or more properties**. For example, you can search specifically for the **Date Created** value on the documents.



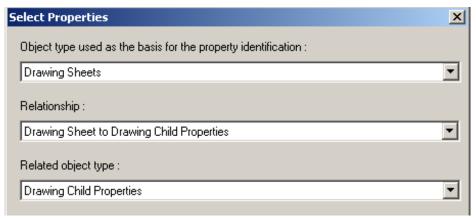
Issue or Revision Properties

To search specifically for issue or revision properties on the drawing sheet or drawing component, set the Relationship to Drawing Sheet (or Drawing Component) to Drawing Issues or Drawing Sheet (or Drawing Component) to Drawing Revisions when setting the filter properties.



Baseline, Style, Smart 3D, or Custom Attribute Properties

To search specifically for baseline, style, Smart 3D, or custom attribute properties, set the **Relationship** to **Drawing Sheet to Drawing Child Properties**, and set **Related object type** to **Drawing Child Properties**.



See Also

Setup Dialog Box (Search Folder) (on page 234)

SECTION 17

Imported Folders

Imported Folder allows you to import an external Windows folder containing any type of file available in Windows. You create an imported folder by right-clicking a folder in the **Management Console** or the **Drawing Console** and selecting **New** > **Imported Folder**.

Imported files are opened using their standard Windows default software. For example, a .docx file opens in Microsoft Word.

Imported Folder Shortcut Menu

Right-click an imported folder component to display the shortcut menu.

Delete

Deletes the imported folder and its files from the database. Deleting does not affect the original folder and files on the network drive.

Rename

Renames the imported folder. Renaming does not affect the files in the folder or the name of the original folder on the network drive.

Revise

Allows you to revise all documents in the imported folder without publishing them. This command is only available if the model is registered with SmartPlant Foundation and when the documents are persisted in the model database. For more information, see *Revise* (on page 421) and *Setup Dialog Box (Imported Folder)* (on page 239).

Export

Exports files in the imported folder to the specified folder path. This command is only available after setup of the imported folder, and when the folder and files are persisted in the database.

Refresh

Refreshes the status of files in the imported folder when the date of a file is earlier than the date of the file the on the network drive. The icon of an out-of-date file is superimposed with \times . This command is only available after setup of the imported folder.

Publish

Publishes a single document or all documents in the search folder. This command is only available if the model is registered with SmartPlant Foundation and when the documents are persisted in the model database. For more information, see *Setup Dialog Box (Imported Folder)* (on page 239).

Update and Publish

Updates and immediately publishes a single document or all documents in the search folder. This command is only available if the model is registered with SmartPlant Foundation and when the documents are persisted in the model database. For more information, see *Setup*

Dialog Box (Imported Folder) (on page 239).

Setup

Specifies the folder to import. For more information, see *Setup (Imported Folder)* (on page 239).

Update Now

Updates all files in the imported folder to match the files on the network drive. This command is only available after setup of the imported folder.

Properties

Displays the properties of the folder.

Imported Files

Right-click an imported file to display the shortcut menu.

Open

Opens the file with the appropriate Windows application.

Properties

Displays the properties of the file.

Update

Updates the file to match the files on the network drive. This command is only available after setup of the imported folder.

★ IMPORTANT Select a shared network folder (with a path beginning with \\) instead of a local folder to allow refresh and update of file changes between users.

NOTE Unlike a folder , you cannot create or paste other components within the imported folder .

Create an imported folder

1. Right click a folder in the **Management Console**, and select **New > Imported Folder**.

The imported folder is added to the hierarchy.

2. Right-click the folder, and select **Setup.**

The **Setup** dialog box displays.

3. Type the path in the **Folder to Import** box.

OR

Click **Browse** , navigate to the needed folder in the **Browse for Folder** dialog box, and click **OK**.

- In the Setup dialog box, click Persist in Database to add the imported files to the model database.
- 5. Click OK.

The imported folder and its files are added to the hierarchy.

6. To rename the folder, right-click the folder, and select **Rename**, or select the folder, and press **F2** on the keyboard. Type a new name.

Setup (Imported Folder)

Sets options for importing an external folder and its files. This command is available on the **Imported Folder** shortcut menu and displays the **Setup** dialog box.

Setup Dialog Box (Imported Folder) (on page 239)

Setup Dialog Box (Imported Folder)

Specifies a folder containing the files to import.

You should import from a shared network folder having a path beginning with \\ to allow everyone to have access to the same files and to avoid multiple file copies or unintended overwriting of changes from another user. Windows handles read-only locking of the files when they are open. You cannot import the %Temp% folder because it is used for file processing during import.

Folder to Import

Specifies the folder path. Type a path, or click **Browse** to select a path.



Allows you to browse for a folder location. For more information, see *Browse for Folder Dialog Box* (on page 239).

Persist in Database

When selected, adds imported files to the model database. When cleared, only the folder path to the files is imported.

NOTE Export is not available when Persist in Database is cleared.

Browse for Folder Dialog Box

Displays a hierarchical list of folders. Select a folder from this list, and click **OK** to choose it, or double-click a folder to display any sub-folders. The selection you make on this dialog box appears in the **Folder to Import** box of the **Setup** dialog box.

Make New Folder

Creates a new folder in the hierarchy, beneath the currently selected folder.

SECTION 18

Tools Menu

Several tools are provided within the Drawings and Reports task. These tools display on the **Tools** menu in the Drawings and Reports task.

Reference Data Tools

The following commands on the **Tools** menu are for use by your reference data administrator. They apply strictly to customizing orthographic drawing and marine mode drawings by rule reference data:

- Define View Style
- Define Layout Style
- Edit Border Template
- Edit Layout Template

You do not use these commands in the normal workflow of creating drawings and reports. For more information, see the *Drawings and Reports Reference Data Guide*.

Other Tools

You can use the **Custom Command** tool to set up special macro commands you use in your documents. **Batch Management** processes drawing updates and printing on a batch server. **Convert Legacy Snapshots** converts legacy version 6.1 snapshot drawings to composed drawings. For more information, see the *Orthographic Drawings User's Guide*.

In This Section

Convert Legacy Snapshots	240
Batch Processing - Intergraph Smart Batch Services	
Custom Commands	

Convert Legacy Snapshots

Converts legacy version 6.1 Snapshot drawings to Composed drawings. As of version 2007, Composed drawings replace Snapshot drawings as a more flexible way to create orthographic drawings. Composed drawings allow you to have views that are managed by a drawing region and associate the views to volumes and other views. If you have legacy Snapshot Drawings in the **Management Console**, they are indicated in the **Detail View** with yellow icons You must convert legacy snapshot drawings to composed drawings in order to edit, change document properties, update, revise, or publish them. To convert legacy snapshot drawing to composed drawings, click **Tools > Convert Legacy Snapshots**.

Convert Legacy Snapshots Dialog Box (on page 241)

After converting the legacy Snapshot drawings, the icon updates, indicating that the drawing is now available for editing and updating as a composed drawing. For more information on composed drawings, see *Composed Drawings* (on page 201). You can open and edit a

composed drawing using the **Tools > Drawings Console** command in any of the other 3D tasks, such as Common or Space Management.

Convert Legacy Snapshots Dialog Box

Shows the **Management Console** hierarchy with legacy version 6.1 Snapshot drawings indicated with yellow icons so you can select and convert them to Composed drawings. You access this dialog box when you select **Tools > Convert Legacy Snapshots**. Select a node to convert all legacy snapshot drawings within the node, or you can select and convert a single drawing document. Click **OK** to convert the selected drawing documents.

After converting the legacy Snapshot drawings, the icon updates, indicating that the drawing is now available for editing and updating as a composed drawing.

Batch Processing - Intergraph Smart Batch Services

With batch processing, you can update, print, or refresh your documents without dedicating your computer to the operation.

Drawings Batch Dialog Box (on page 241)

What do you want to do?

- Add Actions to Queue (on page 242)
- Create a Template (on page 243)

Drawings Batch Dialog Box

Updates, prints, and refreshes document batch jobs without requiring a dedicated computer for the operation.

Available Actions

Lists the actions that you can perform within a batch job. The actions available are based on permissions and document type.

- Refresh Refreshes documents on a batch queue.
- **Full Update** Updates all drawings by rule documents, regardless of the current status of the documents. This action is only available in the **Marine** mode.
- Update Updates documents on a batch queue.
- Save As Specifies the file format to which to save Smart 3D drawings.
- Print Schedules a document to be printed from a batch queue.

Actions to Queue

Lists the actions that are performed in the batch job.

- Remove Removes an action from the queue.
- Clear Removes all actions from the queue.

Action Options

Defines the parameters of an action. The displayed options depend on the selected action.

Template Name

Saves print settings as a template. Type a name in the box, and click **Save Settings as Template.** While optional, creating a new template is a quick method to submit a batch job with the specified options.

Save Settings as Template

Creates a user-parameterized action.

Delete

Removes a template from **Available Actions**. This option is available only on user-created templates.

Save As

For more information on the **Save As** action options, see *Save As Dialog Box* (on page 75).

NOTE In the **Output Folder** field, select a shared folder through its UNC path. A UNC path has the general form of \\server\share.

Print

- Printer Displays all printers configured on the client computer. The name of the printer on the batch queue must match for the print to be successful.
- Copies Specifies the number of copies to print.
- Black and White Prints a black and white drawing, if checked.
- Orientation Indicates the orientation of the printed output. Select Portrait or Landscape.
- Paper Size Displays the paper sizes supported by the selected printer.

Use 64-bit if available

Updates drawings with 64-bit processes. This option is only available for **Update** actions performed on composed drawings.

■ NOTES

- If you select **Use 64-bit if available** and run **Update** on a folder component containing composed drawings and other drawing types, the composed drawings are updated with 64-bit processes. All other drawings are updated with 32-bit processes.
- You cannot update a composed drawing with 64-bit if the drawing contains a view style that includes PDS reference data.

Schedule

Creates the job and opens the **Schedule Batch** dialog box.

■ NOTE If you click **Schedule** with no actions in **Actions to Queue**, you receive a message, and the dialog box remains open.

Add Actions to Queue

You can set up batch jobs for a single document or for multiple documents.

Single document:

- Right-click on a single drawing either in the **Detail View** or in the **Drawings Console**, and select **Batch**.
- 2. Select the action, and click **Add** to move it into **Actions to Queue**.

■ NOTES

- You can add actions to the queue in any order, but the software processes the actions in the following order: Refresh, Update, and then all other actions. If you add Update, but you do not add Refresh, Update is the first action processed.
- You can only add one Save As action to the queue.
- 3. Define any action options. For more information, see *Batch Processing Intergraph Smart Batch Services* (on page 241).
- 4. Click **Schedule** to create the job and open the **Schedule Batch** dialog box.

Multiple documents:

- 1. Do one of the following:
 - Right-click a set of multi-selected documents or components in the **Detail View**, and select **Batch**.
 - b. Right-click a component in the **Management Console**, **Detail View**, or **Drawings Console**, and select **Batch**.
- 2. Select the action, and click Add to move it into Actions to Queue.

■ NOTES

- You can add actions to the queue in any order, but the software processes the actions in the following order: Refresh, Update, and then all other actions. If you add Update, but you do not add Refresh, Update is the first action processed.
- You can only add one Save As action to the gueue.
- 3. Define any action options. For more information, see *Batch Processing Intergraph Smart Batch Services* (on page 241).
- 4. Click Schedule to create the job and open the Schedule Batch dialog box.

Create a Template

Templates are user-parameterized actions that are saved in the session file and available for future batch jobs. You can only create templates based on the **Print** action.

Create a new template:

- 1. Select the **Print** action, and click **Add** to move it to **Actions to Queue**.
- 2. In Action Options, make the necessary changes to the action.
- 3. Type a new **Template Name** for the action.
- Click Save Settings As Template.

Delete a template:

1. Select the template to delete.

2. In Action Options, click Delete.

Schedule [Task] Dialog Box

Queue

Displays the name of the queues configured by an administrator for the job. For more information on configuring the queues, see *Configure Queues for Jobs* in the *Project Management User's Guide*.

Run job

Sets the frequency with which the job runs. Jobs can be scheduled to run once or on a regular interval (daily, weekly, or monthly). Depending on the job frequency selected, additional controls display. These controls allow you to define more specific scheduling information. The scheduling controls can be changed only at job submission.

Run on

Sets the time to start running the job.

Options

Opens the *Optional Schedule Properties Dialog Box* (on page 245) that you can use to define a start and end date.

Run on box

Contains a calendar from which you can select the run date. This option is available when you select **Once** from **Run job**.

Every X days

Specifies how many days pass between job runs. This option is available when you select **Daily** from **Run job**.

Every X weeks

Specifies how many weeks pass between job runs. In addition, you can select on which days the job runs. This option is available when you select **Weekly** from **Run job**.

Day X of the month

Specifies on which day of the month the job runs. This option is available when you select **Monthly** from **Run job**.

The X Y of the month

Specifies on which day of the month the job runs. For example, you can select the last Monday of the month. This option is available when you select **Monthly** from **Run job**.

Job Start

Notifies you when the job starts, if Outlook is set up.

Job Completion

Notifies when the job completes, if Outlook is set up.

Job Abort

Notifies you if the job aborts, if Outlook is set up.

Address Book

Selects the name of the person to be notified by e-mail of the job status, if Outlook is set up. If Outlook is not available, this option does not work. You can also type the address manually. The person you define here receives an email with the job log files after the job finishes.

■ NOTES

- The Batch Services SMTP option must be configured on the batch server for this to work. For more information, see the Intergraph Smart Batch Services documentation.
- The WinZip application is no longer required on the batch server to compress any emailed attachments. Compression is now done with functionality included in Smart 3D.

Optional Schedule Properties Dialog Box

Provides more options on the **Schedule Backup** dialog box. This dialog box opens when you click **Options**.

Start date

Sets an optional start date.

End date

Sets an optional end date, if checked.

Custom Commands

Provides you with application programming capability for the 3D software. Using Microsoft® Visual Basic, you can create a custom command that groups a series of commands and instructions into a single command that runs as an operation in the 3D software. As a result, you can access the customized commands that directly relate to the work routine in your operation.

In Visual Basic, the **Command Wizard** helps you to build a custom command. For example, the first **Command Wizard** step prompts you to identify general information, including command name, project name, author, and company. You can start the wizard in Visual Basic by clicking **Command Wizard** on the **Add-Ins** menu. For more information about installing the **Command Wizard** and other programming resources, see the *Intergraph Smart*TM *3D Installation Guide* available by clicking **Help > Printable Guides** in the software.

After adding a custom command in the 3D software, you can edit it. The **Edit Custom Command** dialog box requires you to specify the program identifier (prog_id), command name and description, command priority, and a command line of arguments in a string.

Delivered Custom Commands

The following list provides descriptions and ProgIDs for the delivered custom commands:

Custom Command	ProgID	Description
Check Database Integrity	SP3DCheckDatabaseIntegrity . CCheckObj	Creates records for the objects that need to be cleaned. You run this custom command directly on a database (Site, Catalog, or Model). After you run this command, you can generate a report to review the errors that the Check Database Integrity command generated. For more information on the Check Database Integrity command, see the Database Integrity Guide available from Help > Printable Guides.
Clean Database	SP3DCleanDatabaseCmd. CCheckObj	Deletes or cleans an object. This command is used when an action on the Check Database Integrity report is To Be Removed or To Be Repaired. For more information on deleting and cleaning objects in the database, see the Database Integrity Guide available from Help > Printable Guides.
Create Drawing View	MenuDrawView. CMenuDrawView	Saves and converts the contents of a three-dimensional graphic view window into a snapshot view. The command creates a rectangular object associated to a clipping volume or volumes in the three-dimensional model.
		Before you create a snapshot view using this command, you must have added at least one composed drawing type to the Management Console in the Drawings and Reports task.
		You can save additional views by updating the view contents and then saving the new design. If you used the Tools > Hide command to avoid displaying certain objects, those objects are included in a composed drawing you create.
		You must have appropriate permissions to access composed drawing types, or you cannot use the Tools > Snapshot View command. If you have only read permission, you receive a message that alerts you to this condition.
		After you create the snapshot views, you can add them to composed drawings when you use the Tools > Drawings Console command.

Custom Command	ProgID	Description
Drawings Check and Repair Utility	DwgCheckUtility,Ingr.SP3D. Drawings.Client.Commands. DwgCheckUtility. RunChecksCmd	Checks drawing items for problems, such as mismatches between views, smartframes, and OIDs; duplications of views, smartframes, and OIDs; and invalid dimensions with missing smartframe attributes. After problems are found, you can run repairs.
		This command is intended for use by your administrator.
Find Object by OID	SP3DFindObjectByReport. FindObjects	Finds objects with integrity problems in a graphic view. Before running this command, you must define your workspace to include these objects. Run a database integrity report, and use the reported OIDs of the objects in the workspace definition. For more information on the Find Objects by OID custom command, see the <i>Database Integrity Guide</i> available from Help > Printable Guides .
Fix Project Root	SP3DPRJMGTRepairCmd. FixCnfgProjectRoot	Synchronizes the model name in the Model database and the Site database. The name in the Site database prevails.
		NOTE You must run this command from a task in the model, not from Project Management.
Fix Sector Size of Documents	DwgBinaryEditorCmd. FixSectorSize	Adjusts the sector size from small to large. Documents that have many sheets and a small sector size can cause the software to run out of memory.
Reset Design Basis Time	IMSEngFrameworkCmd. EngFrameworkCmd Argument = ResetDesignBasisTime	Modifies the Design Basis timestamp. With this command, you can set the time and date back to a point in the past. This command is useful if, for any reason, there are delete instructions that could not be processed.
Synchronize Drawing Component Templates	DwgSynchTemplatesCmd.Sy nchTemplates	Repairs a drawing component that has become corrupted by synchronizing it with a different, uncorrupted drawing component. This command requires that the source component is the same type as the corrupted component, the source component must have a template, and that the source component cannot be corrupted. Also, the number of views on the source component

Custom Command	ProgID	Description
		must be the same as the number of views on the corrupted component. The names of the views on the source component must match the view names of the corrupted component.
Verify P&ID Integrity	SP3DDisplayPIDService. VerifyPIDCmd	Validates the internal connections between objects on a P&ID and objects in the Model database. This command is useful when there is a problem displaying a P&ID or selecting objects on a P&ID. The command provides some basic troubleshooting statistics: Number of design basis objects, number of 3D objects (correlated), number of P&ID objects, number of deleted P&ID OIDs, and number of duplicate OIDs.

Custom Commands Dialog Box (on page 250)

Add Custom Command Dialog Box (on page 250)

Edit Custom Command Dialog Box (on page 251)

What do you want to do?

- Create custom commands (on page 248)
- Add custom commands (on page 249)
- Run a custom command (on page 249)
- Edit a custom command (on page 249)
- Delete a custom command (on page 249)

Create custom commands

1. Open Microsoft® Visual Basic.

TIPS

- You do not create or modify custom commands within the software. You can edit the code of the command in Visual Basic. You can edit a limited number of items, such as the description of the command, using the Edit Custom Command dialog box.
- You must install the Command Wizard software in Visual Basic. The setup for the Command Wizard is located at [Product Folder]\CommonApp\Tools\CommandWizard.
- 2. In Visual Basic, click Add-Ins > Command Wizard.
- 3. Complete all steps on each page of the Command Wizard.

Add custom commands

- 1. Click Tools > Custom Commands.
- 2. On the Custom Commands dialog box, click Add.
- On the Add Custom Command dialog box, type the program identifier you assigned to the command in Microsoft® Visual Basic in the Command ProgID box.
- 4. Type the name you assigned to the command in the **Command name** box.
- 5. Type a phrase that describes the command in the **Description** box.
- 6. If necessary, change the option in the **Priority** section.
- 7. Type command line arguments in a string in the **Argument** box.
- TIP After you complete this procedure, the **Custom Commands** dialog box lists the command you added to the software. You can run the command, edit the settings, or delete the command.

Run a custom command

1. Click Tools > Custom Commands.

The Custom Commands dialog box opens.

- 2. To start a custom command you created, select the command in the list box, and click Run.
- 3. After the command runs, click Close on the Custom Commands dialog box.

Edit a custom command

1. Click Tools > Custom Commands.

The Custom Commands dialog box opens.

- 2. To change the options for a custom command, select the command in the list box, and click **Edit**. For example, you can change the name and description of the command.
- 3. After completing the needed changes, click Close on the Custom Commands dialog box.
- **NOTE** You must open the command in Microsoft® Visual Basic if you want to edit the underlying code.

Delete a custom command

1. Click Tools > Custom Commands.

The Custom Commands dialog box opens.

- 2. Select the command in the list box, and click **Delete**. The software removes the command from the list box; however, the command code is not deleted.
- 3. After completing the needed changes, click Close on the Custom Commands dialog box.
- **NOTE** This action does not delete the DLL for the custom command. It just removes access to the custom command from the **Custom Commands** dialog box.

Custom Commands Dialog Box

Adds and edits customized commands you have created with the **Command Wizard** in Microsoft® Visual Basic. For information on creating custom commands, see *Create custom commands* (on page 248).

Command names

Lists the names of commands that have been added.

Run

Starts the custom command you select in the list box. For more information, see *Run a custom command* (on page 249).

Close

Cancels the Custom Commands dialog box.

Edit

Opens the **Edit Custom Command** dialog box. You can change settings for the command, such as the program identifier (prog_ID) and command name. For more information, see *Edit a custom command* (on page 249).

Add

Installs the custom command into the software. For more information, see *Add custom commands* (on page 249).

Delete

Removes the custom command from the software. For more information, see *Delete a custom command* (on page 249).

Clear

Deletes the information you have typed in the boxes on the **Custom Commands** dialog box.

Description

Contains an identifying phrase so you can better recognize the custom command with which you are working.

Add Custom Command Dialog Box

Accesses a customized command you created in Microsoft® Visual Basic and saves the command within the software.

Command ProgID

Identifies the program identifier for the custom command you created in Visual Basic.

Command name

Specifies the name you assigned to the custom command.

Description

Describes the custom command.

Priority

Assigns a priority of **High**, **Normal**, or **Low**.

Argument

Specifies command line arguments in a string.

See Also

Create custom commands (on page 248)

Edit Custom Command Dialog Box

Changes options for a customized command you added to the software.

Command ProgID

Specifies the program identifier for the custom command you created in Microsoft® Visual Basic.

Command name

Provides a text box for you to change the name you assigned to the custom command.

Description

Provides a text box to provide a descriptive phrase for the custom command.

Priority

Changes priority to High, Normal, or Low.

Argument

Change the command line arguments in a string.

Reset Default

Returns the dialog box to its default settings.

See Also

Create custom commands (on page 248)

Delivered Custom Commands

This section describes some of the delivered custom commands. For a comprehensive list of the custom commands, see *Custom Commands* in the *Common User's Guide*.

Repair Documents Custom Command

The **Repair Documents** custom command updates invalid **Styles.sha** or **Symbol Browser** file paths on a document or a set of documents. Invalid file paths can occur when you change the symbol share. This causes overhead while editing, saving, or updating documents.

Click **Tools > Custom Commands** to add the command using the ProgID **DwgRepairCmd.RepairDocuments**. For more information, see *Custom Commands* in the *Common User's Guide*.

Each time you use **Repair Documents**, the changes made to the component or document are saved in a log file. You can access the file at %TEMP%\Drawings. The log file name starts with

"DwgRepairCmd_" as its prefix. A summary at the end of the log file lists all documents that the command could not repair.

Repair Documents Dialog Box

Component or Document

Specifies the component or document to repair.

Actions

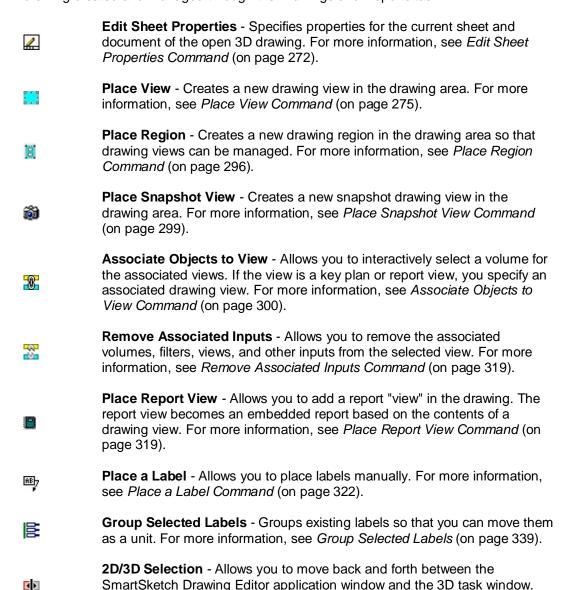
Shows the repair options for the selected component or document.

- Reset style resources Resets the style resource files and to the Styles.sha file on the current symbols share.
- Set symbol browser home Sets the symbol browser home address. You can change
 this address by typing in a new address or by clicking More and opening the file
 folder.

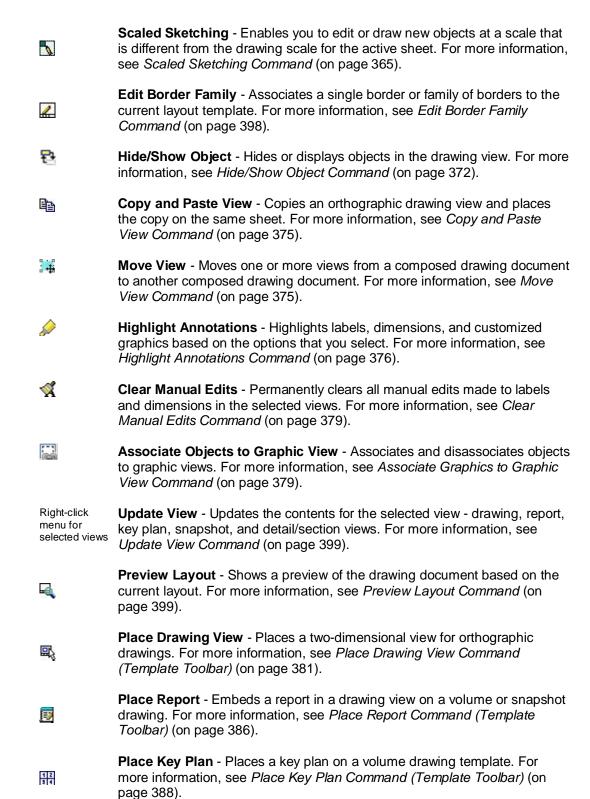
SECTION 19

Working with Drawings and Reports and SmartSketch Drawing Editor

The following commands are available when you are using SmartSketch Drawing Editor to edit a drawing created and managed through the Drawings and Reports task.



For more information, see 2D/3D Selection Command (on page 361).





Place Border Label - Positions drawing property labels in the title block of a template when you are editing a 3D drawing border template. For more information, see Place Drawing Property Label Command (Drawing Labels Toolbar) (on page 391).



Place Drawing Area - Places a drawing area. You use this command to create a drawing area on a 3D drawing border template that has been imported from other software, such as MicroStation DGN or AutoCAD DWG. For information on this command, see Place Drawing Area Command (on page 396). For more information on editing border templates, see the Drawings and Reports User's Guide.

Drawings View Explorer

The Drawings View Explorer provides access to views and their attributes for the currently displayed drawing or template. For more information, see Drawings View Explorer (on page 256).

Dimensioning in 3D Drawings

You can use the Dimension toolbar to place dimensions on your 3D drawings. For more information, see *Dimensioning in 3D Drawings* (on page 267).

You can also specify dimensioning for paper space objects (at the proper scale) when they are drawn on top of drawing view objects. For more information, see Dimension Paper Space Objects for 3D Drawings (on page 268).

Cutting Planes, Detail Envelopes, and Section/Detail Views

The following commands allow you to create and edit cutting planes, detail envelopes, and section and detail views in 3D drawings.



Place Detail Envelope - Creates a detail view for an existing drawing view. Detail views are more than enlargements of the main drawing view. They often contain additional graphical information that is not visible in the main drawing view, such as weld or chalk information. For more information, see Place Detail Envelope Command (on page 343).



Place Cutting Plane - Creates a cutting plane on a drawing view. The cutting plane is a marker that indicates where to slice a group of objects and from which direction to look at that slice. For more information, see Place Cutting Plane/Section View Command (on page 346).



Place Section/Detail View - Creates both a section view and a detail view based on the selected cutting plane or detail envelope. A detail view is extracted from a main drawing view or another detail view. You can rotate detail views in 2D space, but they remain in the same orientation as the main drawing view. Section views are extracted from main drawing views, detail views, or other section views. Section views are similar to detail views, except that they can display information in an orientation that is different from that of the originating view. For more information, see *Place Detail View Command* (on page 358).

Converting Excel Spreadsheet Reports to Native Text Boxes

The **SP3DConvertExcelEmbedded.dll** is a delivered custom command that allows you to convert an Excel spreadsheet report to the native text box format for use in 3D Drawings. For more information on converting Excel spreadsheet reports, see *Convert Excel Spreadsheet Reports to Native Text Box Format Custom Command* (on page 403).

Saving Drawings in MicroStation and AutoCAD Format

You can use the Save As command to assign dimension units and export the drawings to MicroStation or AutoCAD format. For more information, see Save As MicroStation or AutoCAD Format (on page 270).

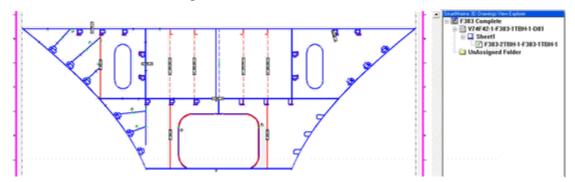
Drawings View Explorer

Provides access to views and their attributes for the currently displayed drawing or template.

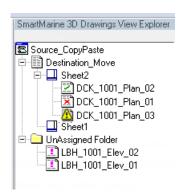
Drawings View Explorer

Displays a list of sheets and views on the active sheet and a list of unplaced views available for placement. This window is available in SmartSketch Drawing Editor when you have a Drawings by Rule template or drawing open. The **Drawings View Explorer** displays by default as a right-side window in SmartSketch Drawing Editor when you:

- Edit a template associated with a Drawings by Rule component. Right-click the component, and select **Edit Template** on the shortcut menu.
- Edit an existing drawing document. Right-click a Drawings by Rule drawing document in the Detail View or the Drawing Console, and select Edit on the shortcut menu.



The **Drawings View Explorer** allows you to see views that are available for the current drawing. Assigned views appear beneath a drawing sheet. Unassigned views appear in the **UnAssigned Folder**. Views are identified by the following icons:



- The view is assigned to a drawing and all 2D geometry is up-to-date with the 3D model.
- The view is assigned to a drawing and 2D geometry is out-of-date with the 3D model.
- The view is not assigned to a drawing, and is in the **UnAssigned Folder**.
- The view is assigned to a drawing, and there were errors in processing the 2D geometry. The update skips the geometry errors.

Shortcut Menu

The **Drawings View Explorer** also allows you to update, refresh, delete, or edit properties by right-clicking on a view or a set of selected views from the same sheet.

Update

Performs a smart update of geometry in the view(s). Rules in the software determine whether to perform:

- An incremental update of the geometry for added, modified, and deleted objects.
- A full update of all geometry.

For more information, see Update and Full Update Commands (on page 414).

NOTE When **Update** performs the first option, it is usually faster than **Full Update** but still results in all geometry being up-to-date for the view.

Full Update

Performs a full update of all geometry in the view. For more information, see *Update and Full Update Commands (on page 414).*

Refresh

Compares the date of the last update of the views with the modification date in the model for any object that has a *positive* (can be seen) resymbolization in the drawing. For more information, see *Refresh* in the *Orthographic Drawings User's Guide*.

Delete

Moves one or more views to the **UnAssigned Folder**, or deletes views permanently if the view was placed by the drawing rule set. For more information, see *Place View Command* (on page 275).

Properties

Opens the **Drawing View Properties** dialog box, allowing you to change the properties of a drawing view. For more information, see *Drawing View Properties Dialog Box (Place View Command) - Steel Order Drawings* (on page 281).

Update Selected

Performs an update on objects that are first selected in a view or in the 3D model. You can perform **Update Selected** on both full and cropped views. Only the selected geometry is

updated in the view. Other geometry affected by the selected geometry is neither checked

nor updated. As a result, the view is marked as out-of-date . This option is only available when:

- A drawing is created with a Drawings by Rule component. For more information, see the Orthographic Drawings User's Guide.
- 2D/3D Selection is clicked. 2D/3D Selection causes geometry in the view to be actively linked to its 3D model object. For more information, see 2D/3D Selection Command (on page 361) in the SmartSketch Drawing Editor User's Guide.

View Log

Displays a log of the results from the last update performed on the view.

■ NOTES

- You can also select multiple views from the RAD sheet.
- For more information, see Place an Unassigned View (on page 408).

Move Sheet(s)

Moves the selected sheets from one document to another under the same Drawings by Rule component. For more information, see *Move Sheet(s)* (on page 258).

Move Sheet(s)

Moves the selected sheets from one document to another under the same Drawings by Rule component.

1. In the Drawings View Explorer, right-click the sheets to be moved, and select **Move Sheet(s)**.

The Move Sheet dialog box displays.

2. Select the destination document, and click **OK**.

NOTE Sheets cannot be moved to a document that is being edited or to a document on which you do not have write permissions.

The sheets are moved to the destination document.

■ NOTES

- You can only move sheets to documents under the active component.
- When all sheets under a document are moved, an empty sheet with a border template is placed under the source document.
- To perform **Move Sheet(s)**, the component must have at least two documents as children.
- Move Sheet(s) cannot be used by selecting both views and sheets together.
- Upon completion, the status of the source and destination documents is changed to "Out of date."

Dimensions

In general, dimension rules control the placement of dimensions, and dimension styles control the appearance, including the units, of dimensions in orthographic drawings. However, dimension styles and dimension rules interact in complex ways. There are two methods you can use to cause dimensions to display in drawings. *Automatic dimensioning* and *manual dimensioning* place dimensions in native format drawings.

For *automatic dimensioning*, the view style controls whether or not dimensions are placed. For *manual dimensioning*, you edit an existing drawing and place dimensions manually.

NOTE The **Save As** command saves drawings from the database to files and presents additional considerations about dimensions. The files created by the **Save As** command can be native format or a foreign format such as MicroStation or AutoCAD. With the **Save As** command, the software attempts to replicate dimensions as they are shown within a drawing.

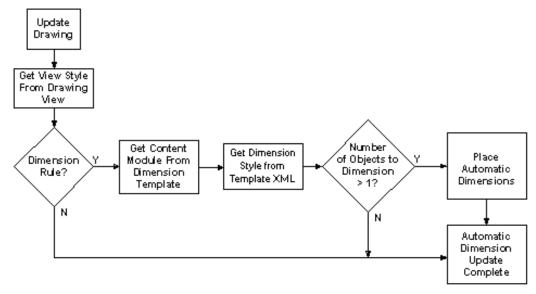
Assigning dimension units is different for each method. For more information, see the following sections in the Orthographic Drawings User's Guide:

Automatic Dimensioning (on page 259)
Manual Dimensioning (on page 266)
Save As MicroStation or AutoCAD Format (on page 270)

■ NOTE Isometric drawings use the isometric options settings within their style to determine the dimension appearance, placement, and units. For more information, see *Isometric Drawing Styles* in the *Isogen Isometric Drawings User's Guide*.

Automatic Dimensioning

When you use automatic dimensioning, the view style controls whether or not dimensions are placed. The following flowchart shows how **Update** (for marine mode Drawings by Rule) and **Update Now** (for all other drawing types) assign units to automatically placed dimensions.



View Styles

A dimension rule within a view style triggers the automatic placement of dimensions in an orthographic drawing. For more information, see *Use View Styles with Dimension Rules* (on page 260).

Dimension Rules

Dimension rules control the placement and appearance of automatic dimensions in the drawing. For more information, see *Use Dimension Rules* (on page 261).

Dimension Templates

The dimension rule is not responsible for assigning the dimension units to automatic dimensions. The dimension rule points to a dimension template XML file that influences the unit assignment. For more information, see *Use Dimension Templates* (on page 261).

Dimension Styles

You can determine the active style of a dimension by editing the drawing or drawing template in **SmartSketch Drawing Editor**. For more information, see *Edit Dimension Styles* (on page 263).

Dimension Style Overrides

You can override dimension style settings in **SmartSketch Drawing Editor**. For more information, see *Override Dimension Styles* (on page 265).

See Also

Dimensions (on page 259)

Use View Styles with Dimension Rules

A dimension rule within a view style triggers the automatic placement of dimensions in a drawing.

Select Tools > Define View Style.

The **Define View Style** dialog box displays.

- 2. Select **New Style** to define a new view style with a dimension rule. You can also edit an existing view style by selecting **Properties**.
- 3. On the View Style Properties dialog box, use the Dimension Rule field dropdown to specify a dimension rule for the view style. For more information on this dialog box, see View Style Properties Dialog Box in the Drawings and Reports Reference Data Guide. Select More to display the Select Dimension Rule dialog box. For more information, see Select Dimension Rule Dialog Box in the Drawings and Reports Reference Data Guide.



For information on how dimension rules are maintained, see *Use Dimension Rules* (on page 261).

▶ NOTE For more information on defining view styles, see *Define View Style Command* in the *Drawings and Reports Reference Data Guide*, accessible using the **Help > Printable Guides** command in the Drawings and Reports task.

See Also

Automatic Dimensioning (on page 259)
Dimensions (on page 259)

Use Dimension Rules

A dimension rule controls the placement and appearance of automatic dimension in the drawing. When you select **More** in the **Dimension Rule** dropdown on the **View Style Properties** dialog box, the **Select Dimension Rule** dialog box displays the list of rules available in the Drawings catalog. For more information, see *Select Dimension Rule Dialog Box* in the *Drawings and Reports Reference Data Guide*. You can access this guide with the **Help > Printable Guides** command in the Drawings and Reports task.

The Drawings catalog is file-based and located on the SharedContent share within the \Drawings\Catalog folder. The available dimension rules are XML files stored in the \Drawings\Catalog\Rules\DimensionRules folder. Several example dimension rules are delivered with the software.

You edit the dimension rule XML files with a text or XML editor. Each dimension rule should point to its own dimension template. Rename any customized dimension rules. Do not use the delivered rule names for customized rules.

★ IMPORTANT

- We recommend that you maintain a separate set of dimension rules for each dimension style used in drawings.
- The dimension rule is not responsible for assigning the dimension units to automatic dimensions. However, the dimension rule points to another XML file, called the *dimension template*, that influences the unit assignment. For more information, see *Use Dimension Templates* (on page 261).
- The drawing dimension XML Files are discussed further in the *Intergraph Smart*TM 3D *Programmer's Guide* under *Extending the Capabilities of the Software*. Contact your administrator or Intergraph Support if you need the *Intergraph Smart*TM 3D *Programmer's Guide*. You can find support information on our web site *http://support.intergraph.com* (http://support.intergraph.com/).

See Also

Automatic Dimensioning (on page 259) Dimensions (on page 259)

Use Dimension Templates

The dimension rule is not responsible for assigning the dimension units to automatic dimensions. However, the dimension rule points to another XML file, called the *dimension template*, that influences the unit assignment. The dimension template XML file contains the settings that further control the placement and appearance of dimension in the orthographic drawings.

You edit the dimension template XML files with a text or XML editor. Rename any customized dimension templates. Do not use the delivered rule names for customized dimension templates.

Several example dimension template XML files are delivered with the software and are located on the SharedContent share in the \Drawings\Catalog\Dimensions\Templates folder.

The setting in the template that influences dimension unit display is <dimensionContentModules>. In each dimension template, you should set <dimensionContentModules> to the dimension style used to create dimensions in the drawing. The values available for this setting determine whether the software places the dimension vertical, horizontal, radial, or angular. To determine the dimension units, the various content modules refer to the dimension formatting saved either in the drawing template file (for volume and by query drawing types) or in the drawing itself (for composed drawings).

★ IMPORTANT

- We recommend that you maintain a separate dimension template for each dimension rule you create.
- For manual dimensioning, the software looks in the Linear_A_HV dimension template for the value of the dimension content module during the update of manually placed dimensions. For more information, see Manual Dimensioning (on page 266).
- The <dimensionStyleSettings> setting in the dimension templates is not used by the software at this time. However, when creating drawings, we recommend that the active dimension style in the selected border template have the same name as the <dimensionStyleSettings> setting in the dimension template to avoid problems in future software releases. Rename any customized border templates. Do not use the delivered border template name for customized border templates.
- The drawing dimension XML Files are discussed further in the *Intergraph Smart*TM 3D *Programmer's Guide* under *Extending the Capabilities of the Software*. Contact your administrator or Intergraph Support if you need the *Intergraph Smart*TM 3D *Programmer's Guide*. You can find support information on our web site *http://support.intergraph.com* (http://support.intergraph.com/).

Overall Dimensions

By default, dimension control generators use the settings in the Linear_A_HV.xml dimension template. By using **<overallDimension>**, you can override Linear_A_HV.xml and select another template for overall dimensions. In the example below, Piping Plan_Pipes_Horizontal.xml is used as the dimension template for piping parts:

```
<dimensionSettings>
    <overall>-1</overall>
    <overallDimension value="Piping Plan_Pipes_Horizontal">
</dimensionSettings>
```

Layering Dimensions

You can define the layer on which your dimensions display in the drawing by editing the dimension rule template XML file. After the **<dimensionStyleSettings>** section of the XML file, add a **<dimensionLayerSettings>** definition using the **<dimLayer>** tag, like the one shown in the example below, that defines the layer on which you want the dimensions placed:

```
<dimensionLayerSettings>
     <dimLayer>dimensionLayer</dimLayer>
</dimensionLayerSettings>
```

If the layer does not already exist in the template you are using to create drawings, the software creates the layer automatically.

See Also

Automatic Dimensioning (on page 259)
Dimensions (on page 259)
Use Dimension Rules (on page 261)

Edit Dimension Styles

The software resolves dimension units for a particular drawing from the active dimension formatting in either the drawing template (for volume and *by query* drawing types) or the drawing itself (for composed drawings). The active dimension formatting is a combination of the settings in the active dimension style plus any dimension style overrides that may be set.

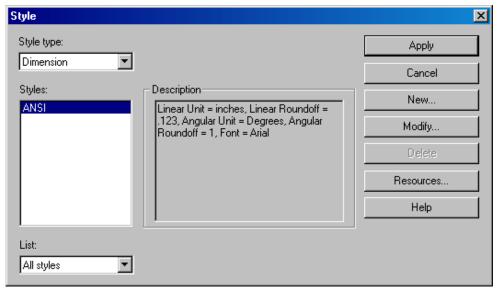
You can determine the active style by editing the drawing or the drawing template in **SmartSketch Drawing Editor**, selecting one of the placement dimension commands, and checking the style that displays on the resulting ribbon bar. You can view the overrides to the active dimension in the drawing template by selecting **Format > Dimension** in **SmartSketch Drawing Editor**.

★ IMPORTANT The Imperial border templates have ANSI as the active dimension style with no overrides set. The Metric border templates have DIN as the active dimension style with no overrides set.

You can create and maintain dimension styles in SmartSketch Drawing Editor.

1. In SmartSketch Drawing Editor, select Format > Style.

The Style dialog box displays.



2. Click Modify.

The Modify Dimension Style dialog box displays.

3. Modify the appearance of the dimension style as needed, and click **OK**.

4. On the **Style** dialog box, you can either save dimension styles with the files being edited or reference them from other files. You can reference files by selecting **Resources**.

TIPS

- The reference files, also called resource files, can be any file with an .igr or .sha extension. While referenced dimension styles are available for dimension placement, they cannot be edited. Therefore, the Style dialog box only displays dimension styles that are saved locally to the file.
- In cases where a local dimension style has the same name as a referenced dimension style, the software always uses the local style when placing dimensions. Currently, you cannot override local dimension styles with the same name as a referenced dimension style of the same name. Local dimension styles can be renamed so that the referenced dimension style can be used to place dimensions. Local dimension styles are renamed on the **Modify Dimension Style** dialog box. For more information on overriding dimension style settings, see Override Dimension Styles (on page 265).

The delivered *Styles.sha* file contains several example dimension styles. This file is located in the Symbols share in the *\Drawings\Catalog\Templates* folder. All drawings created in the Drawings and Reports task reference the *Styles.sha* file.

★ IMPORTANT After updating the drawings, the *Styles.sha* file on the Symbols share is the only file referenced into the drawings. This is true for all orthographic drawings. Any other files previously referenced before the update need to be referenced to the drawing again.

All drawings and drawing templates must have at least one dimension style saved within them. Therefore, if only one dimension style is saved with a file, you cannot delete it. Also, you cannot delete any style from a drawing or drawing template if they are currently used by dimensions in a drawing.

Dimension styles can be added to the list of saved or local styles in a drawing or drawing template in one of two ways:

- Create a new dimension style from the Style dialog box using the New... command.
- Place a dimension in a drawing or drawing template using a reference dimension style. This
 method copies the referenced dimension style into the drawing or drawing template.

★ IMPORTANT

- We recommend that you maintain a separate set of border templates for each dimension style used for drawings. The border templates within a particular set should have the same active dimension style with the same style settings.
- The <dimensionStyleSettings> setting in the dimension templates is not used by the software at this time. However, when creating drawings, we recommend that the active dimension style in the selected border template have the same name as the <dimensionStyleSettings> setting in the dimension template to avoid problems in future software releases. Rename any customized border templates. Do not use the delivered border template name for customized border templates.
- We recommend that you maintain a complete set of your dimension styles in a renamed version of the Styles.sha file.

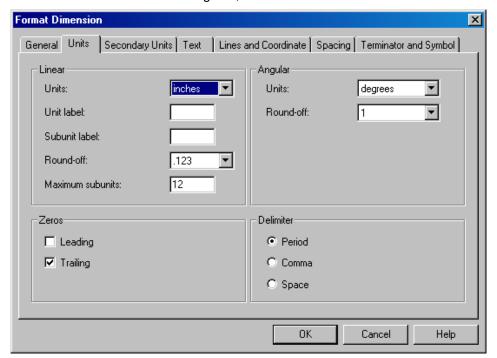
See Also

Override Dimension Styles (on page 265)

Override Dimension Styles

You can override dimension styles in a drawing or drawing template.

- ★ IMPORTANT We support dimension style overrides, but we do not recommend their use as a standard practice. Overrides are saved in the drawing or drawing template and can be difficult to maintain as dimension styles change. As an alternative, we recommend creating a new dimension style in a reference resource file. For more information, see *Edit Dimension Styles* (on page 263).
- Select Tools > Edit Border Template, and select a template to edit.
 The template opens in SmartSketch Drawing Editor.
- 2. Select Format > Dimension.
- 3. On the **Format Dimension** dialog box, select the **Units** tab.



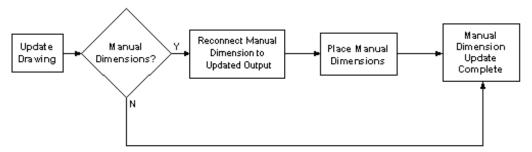
- 4. If the settings on the **Units** tab are identical to those on the **Modify Dimension Style** dialog box, no overrides are applied to the active dimension style. If a dimension style is changed during a dimension placement command, the override values become identical to the dimension style values. For more information, see *Edit Dimension Styles* (on page 263).
- NOTE You cannot override local dimension styles with the same name as a referenced dimension style. Local dimension styles can be renamed so that the referenced dimension style can be used to place dimensions. Local dimension styles are renamed on the Modify Dimension Style dialog box.

See Also

Automatic Dimensioning (on page 259) Dimensions (on page 259) Use Dimension Rules (on page 261)

Manual Dimensioning

When you use *manual dimensioning*, the view style and the dimension rules do not apply for display or placement. However, the logic used to determine manual dimensions is similar to that of *automatic dimensions*. The following flowchart shows how **Update** (for marine mode Drawings by Rule) and **Update Now** (for all other drawing types) assign units to manually placed dimensions.



Dimension templates

For *manual dimensioning*, the software looks in the dimension template for the value of the dimension content module during the update of manually placed dimensions. For more information, see *Use Dimension Templates* (on page 261).

Dimension styles

You can determine the active style of a dimension by editing the drawing or drawing template in **SmartSketch Drawing Editor**. For more information, see *Edit Dimension Styles* (on page 263).

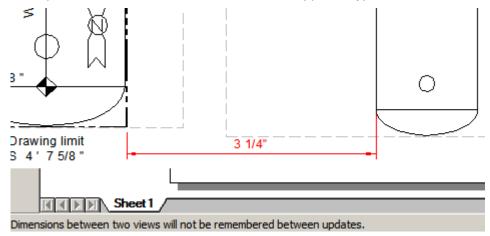
Dimension style overrides

You can override dimension style settings in **SmartSketch Drawing Editor**. For more information, see *Override Dimension Styles* (on page 265).

NOTE If you place a manual linear dimension that is chained or stacked with an automatic linear dimension, the manual dimension is immediately replaced with a standalone dimension that is no longer attached to the automatic dimension. The new dimension is attached to a newly-placed point object whose position coincides with the connect point location of the automatic dimension projection line to which the manual dimension was originally attached. Because the new dimension is no longer chained or stacked with the automatic dimension, it does not move if the automatic dimension's position is modified.

Dimension between views

If you try to place dimensions from one drawing view to another, a status bar message displays **Dimensions between two views will not be remembered between updates.** The dimension is not placed and the command cancels. Other supported types of dimensions remain.



Dimensioning in 3D Drawings

When you edit a Smart 3D drawing in SmartSketch Drawing Editor, you can use commands on the **Dimension** toolbar to manually place dimensions. When updating the drawing, the software remembers dimensions placed between objects within a single drawing view, dimensions placed from paper space to paper space, and dimensions placed between paper space graphics and model objects. You cannot dimension from one drawing view to another drawing view.

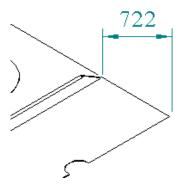
You can place dimensions in drawings by rule and composed drawing views using the standard SmartSketch Drawing Editor dimensioning commands. The commands are enhanced with an additional ribbon to support accurate dimensions of 3D objects in non-planar views.

Dimensioning Drawing Elements in the SmartSketch Drawing Editor Help

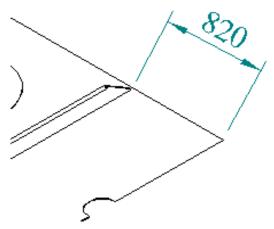
Dimension Ribbon in the SmartSketch Drawing Editor Help

3D Dimension Ribbon

The 2D dimension commands treat all geometry in a view as being in the view plane. If the view contains non-planar geometry, such as in an isometric view, the value of the dimension is not accurate, as shown in the following figure.



With the 3D dimension options, the dimension reflects the distance between the 3D objects.



You can specify dimensioning for paper space objects (at the proper scale) when they are drawn on top of drawing view objects. For more information, see *Dimension Paper Space Objects for 3D Drawings* (on page 268).

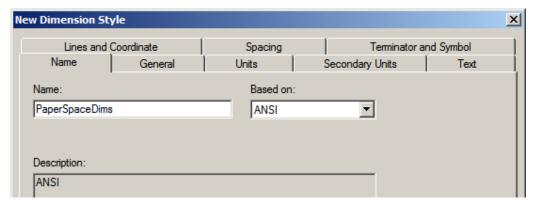
Dimension Paper Space Objects for 3D Drawings

The following procedure shows you how to dimension paper space objects (at the proper scale) when they are drawn on top of a 3D Drawing view object. To accomplish this task, you create a dimension style with the appropriate scale setting in SmartSketch Drawing Editor.

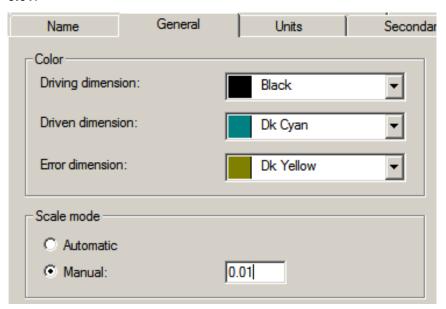
When manually placing dimensions and then updating the drawing, the software remembers dimensions placed between objects within a single drawing view, dimensions placed from paper space to paper space, and dimensions placed between paper space graphics and model objects.

- ★ IMPORTANT Dimension styles must be defined for paper space objects and their scale factors, otherwise the paper space dimensions will be deleted when the drawing is updated. SmartSketch Drawing Editor will not recognize any possible errors in the dimension style, so be sure that any new dimension styles are correct before updating a drawing. For more information, see Edit Dimension Styles (on page 263) and Manual Dimensioning (on page 266).
- 1. Start SmartSketch Drawing Editor. Double-click shape2dserver.exe in the [Product Folder]\Common2D\Shape2D\Bin folder.
- Use File > Open to open the styles.sha file. This file is located in your [Product Folder\Common2D\Shape2D\Template\Styles folder.
 - **NOTE** The *styles.sha* file is delivered as a read-only file. You can change the properties of the file, or you can copy the file to a new name and use the copied file to create your new dimensioning style.
- 3. Select Format > Style, and on the Style dialog box, click New.

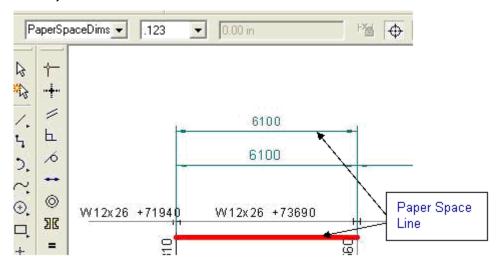
4. On the New Dimension Style dialog box, give the new paper space dimension a name.



5. Go to the **General** tab, and set the scale mode to **Manual** and set a manual value for the dimensioning. For example, for a scale of **1:100mm**, you would set the manual value to **0.01**.



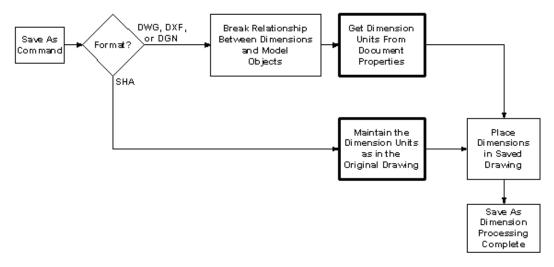
When you apply this updated *style.sha* to your drawings, you can use the new paper space dimension style when placing paper space objects, even when positioned on top of drawing view objects.



- ★IMPORTANT Upon resizing a drawing, all automatic dimensions change accordingly, but manual dimensions will not relocate. Manual dimensions between paper space objects and model objects will still exist, but their values will change.
- **NOTE** Dimensioning from model objects in one view to model objects in a different view is not supported.

Save As MicroStation or AutoCAD Format

You can use the **Save As** command to assign dimension units and export the drawings to MicroStation or AutoCAD format. The following flowchart shows how the **Save As** command controls dimensioning during export.



Save As Command

The **Save As** command is available from the shortcut menu for any document or node containing documents in the **Console**. For orthographic drawings, the **Save As** command supports exporting to DGN, DWG, and DXF formats, as well as the native SHA formats. For more information, see *Save As Command* (on page 70) in the *Drawings Help*.

NOTE For Isogen Isometric Drawings, a file is created for each sheet in the drawing with [drawing name]_[sheet name] as the filename. For example, if the drawing My_Pipeline contains Sheet1 and Sheet2, two files will be saved with the names My_Pipeline_Sheet1 and My_Pipeline_Sheet2.

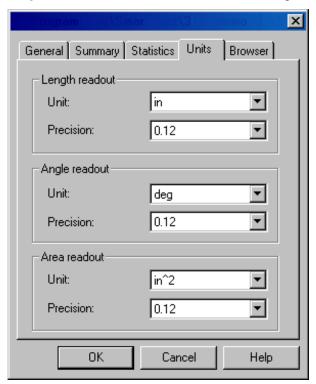
Embedded Object Dimensioning

The model graphics in the drawings are embedded in the drawing file. The dimensions in the drawings are *connected* directly to the embedded objects.

During export to MicroStation or AutoCAD formats using **Save As**, the software opens the drawings in **SmartSketch Drawing Editor** and the relationship between the embedded objects and the dimensions is broken. The dimension, however, is not removed and still maintains its original value.

Document Properties

Dimensions in drawings exported to the DGN, DWG, and DXF formats do not use the active dimension formatting for their units. Instead, the software determines dimension units from the document properties for the drawing being exported. This behavior applies whether the drawings is a volume, *by query*, or composed drawing type. You can view and edit the document units by opening the drawing in **SmartSketch Drawing Editor** and selecting **File > Properties**. You can see the current unit settings on the **Units** tab.



★ IMPORTANT We recommend that border templates have identical units and precision values in the document properties set in the active dimension style. This maintains dimension unit accuracy if you export the drawing to a foreign file format.

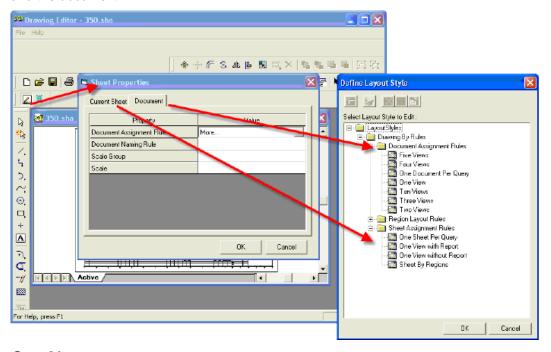
See Also

Dimensions (on page 259)

Edit Sheet Properties Command

- Specifies properties for the current sheet and document. This command is available on the Drawings Compose toolbar in SmartSketch Drawing Editor when you:
- Edit a template associated with a Drawings by Rule component. Right-click the component and select **Edit Template** on the shortcut menu.
- Edit an existing drawing document. Right-click a Drawings by Rule drawing document in the
 Detail View and select Edit on the shortcut menu.

In both cases, the template or drawing document opens in SmartSketch Drawing Editor with the **Edit Sheet Properties** available on the toolbar above the drawing area. When you click the command, the **Sheet Properties** dialog box appears so you can set the properties for the sheet and the document.



See Also

Sheet Properties Dialog Box (on page 273)

Change the Border Template for an Existing Sheet

In this workflow, you change the border for an individual sheet to a template different from the border template defined in the component. For example, you may want the first sheet of drawing to use a different border template than all other sheets.

- Right-click the drawing document in the Management Console Detail View, or in the Drawing Console and select Edit. The drawing opens in SmartSketch Drawing Editor.
- 2. Select a sheet tab at the bottom of the drawing view.
- 3. Click Edit Sheet Properties an on the toolbar. The Sheet Properties dialog box displays.
- 4. On the **Current Sheet** tab, select a new value for **Border Template**.
- 5. Click **OK**. The new border template is applied to the sheet.

■ NOTES

- Layout Template is inactive, and cannot be changed for an existing sheet
- Border Layout and Template Layout are customizable items.
- For more information about layout properties, see *Edit Sheet Properties Command* (on page 272) in *SmartSketch Drawing Editor Help*.

Sheet Properties Dialog Box

Specifies properties for the current sheet and document. This dialog box displays when you click **Edit Sheet Properties** , which is only available in **SmartSketch Drawing Editor** when you are editing a Drawings by Rule template or document.

See Also

Edit Sheet Properties Command (on page 272) Current Sheet Tab (on page 274) Document Tab (on page 273)

Document Tab

Specifies properties for the document. This dialog box displays when you click **Edit Sheet Properties** , which is only available in **SmartSketch Drawing Editor** when you are editing a Drawings by Ruleset template or document.

The properties available are defined as follows:

Document Assignment Rule

Specifies the layout style rule to use for the document. Select **More** in the dropdown to display the **Define Layout Style** dialog box and select the layout style to use.

Document Naming Rule

Defines how the document will be named.

Scale Group

Specifies a scale group for the document, such as Metric, Imperial, or Manual Scale. When you select a scale group, the **Scale** property values update. Selecting **Manual Scale** shows all scales, metric or imperial.

Scale

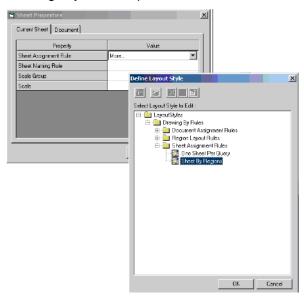
Specifies a scale to use for the document. The **Scale** values update depending on the selected **Scale Group**.

See Also

Edit Sheet Properties Command (on page 272) Sheet Properties Dialog Box (on page 273)

Current Sheet Tab

Specifies properties for the current sheet. This dialog box displays when you click **Edit Sheet Properties** , which is only available in **SmartSketch Drawing Editor** when you are editing a Drawings by Rule template or document.



The properties available are defined as follows:

Sheet Assignment Rule

Specifies the layout style rule to use for the sheet. Select **More** in the list to display the **Define Layout Style** dialog box and select the layout style to use.

Sheet Naming Rule

Defines how the sheet will be named.

Scale Group

Specifies a scale group for the sheet, such as Metric, Imperial, or Manual Scale. When you select a scale group, the **Scale** property values update. Selecting **Manual Scale** shows all scales, metric or imperial.

Scale

Specifies a scale to use for the sheet. The **Scale** values update depending on the selected **Scale Group**.

See Also

Edit Sheet Properties Command (on page 272) Sheet Properties Dialog Box (on page 273)

Drawings Compose Toolbar

This toolbar is available in SmartSketch Drawing Editor when you click **Edit** on a drawing document shortcut menu in the Drawings and Reports task. For more information, see *Drawing Document Shortcut Menu* (on page 36) and *Edit Command* (on page 40) in the *Drawings and Reports Reference Data Guide*.

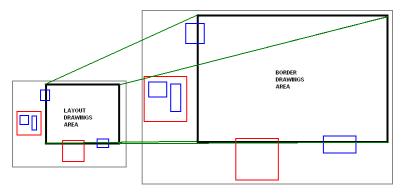
Place View Command

Creates a new drawing view in the drawing area. This command is only available in SmartSketch Drawing Editor when you create a new drawing or open an existing drawing from a 3D task.

When you click **Place View**, you click and drag to define the new view. The **Drawing View Properties** dialog box appears so that you can define the properties associated with the view. The properties shown change depending on the View Style type you select, such as Orthographic View or Key Plan View.

After creating the view and defining its properties, use the **Associate Objects to View** command to specify the content of the view. You can also move the view around, crop it, or align it in the drawing sheet.

■ NOTE You can create drawing views outside the drawing area. Also, when you stretch the drawing area to match the border file, the software retains your offsets so that a view or region outside the drawing area remains outside the drawing area even after it is placed in the border file.



You can create a drawing view outside a region, but this makes the drawing view an "unmanaged view," meaning the properties of the region do not impact the drawing view. However, if a view is entirely inside a region or touching a region, the region *manages* the drawing view and, when you update the drawing in a 3D task, the software pulls the drawing view into the region and updates it based on the region layout style.

See Also

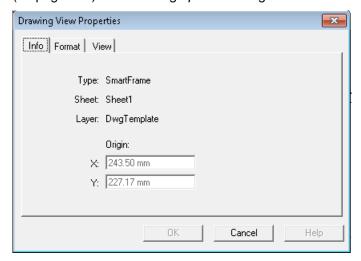
Drawing View Properties Dialog Box (Place View Command) - Composed Drawings (on page 276)

Update View Command (on page 399)

Drawing View Properties Dialog Box (Place View Command) - Composed Drawings

Specifies properties for the selected drawing view in a composed drawing. You can access this dialog box when you right-click on a drawing view and then select **Properties** on the shortcut menu.

For more information on creating a composed drawing, see *Create a new composed drawing* (on page 205) in the *Orthographic Drawings User's Guide*.



Info Tab (Drawing View Properties Dialog Box) (on page 276)

Format Tab (Drawing View Properties Dialog Box) (on page 277)

View Tab (Drawing View Properties Dialog Box) (on page 277)

See Also

Place View Command (on page 275)

Info Tab (Drawing View Properties Dialog Box)

Specifies general view properties.

Type

Displays the category of the selected element.

Sheet

Displays the name of the drawing sheet that contains the selected element.

Layer

Shows the layer that contains the selected element.

Origin

Specifies the coordinates, or location, of an element along the X- and Y-axes.

Format Tab (Drawing View Properties Dialog Box)

Formats the frame around a drawing view.

Show Border

Displays the frame around the object.

Color

Sets the color of the frame.

Line Width

Sets the line thickness on the frame.

Line Type

Overrides a line type for a drawing sheet or embedded object and sets another line style for an element or a linked object.

View Tab (Drawing View Properties Dialog Box)

Sets the drawing view style and other properties for a selected drawing view in a composed drawing.

Style

Specifies a view style, which includes rules for filters, updates, and graphics. The view style controls the output characteristics of the view on the generated drawing. The list displays the 10 most recently used view styles in the session. Click **More...** to display the **Select View Style** dialog box.

Name

Specifies a name for the view. You must type a name in order to create a view.

View Style

Identifies the view style used within the rule set associated with this drawing document. This is a read-only field and is shown for Ruleset view styles only.

Coordinate System Properties

Annotation Coordinate System

Specifies the coordinate system used to place annotations (labels and dimensions) on the drawing. This property is not supported in Ruleset view styles. Choose a coordinate system from the list, or click **More...** to choose another coordinate system with the **Select System** dialog box.

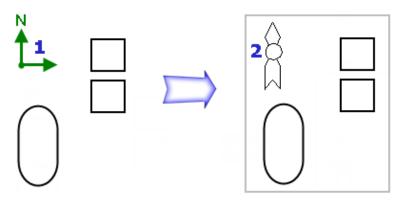
View Direction Coordinate System

Specifies the coordinate system to use with regard to the "looking direction" for the drawing view contents. The coordinates listed are defined for the model. This property is not supported in Ruleset view styles. Choose a coordinate system from the list, or click **More...** to choose another coordinate system with the **Select System** dialog box.

Example 1

The global coordinate system is selected for both Annotation Coordinate System and

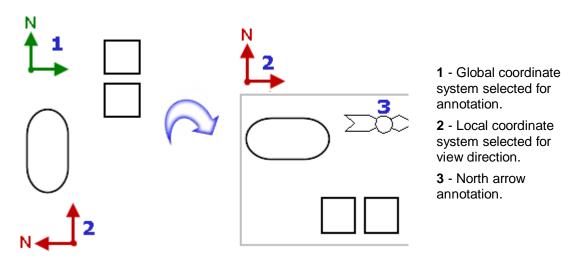
View Direction Coordinate System. A plan view is created, and the global view direction coordinate system North points up. Because the North arrow annotation is also defined by the global coordinate system, the North arrow annotation also points up.



- 1 Global coordinate system selected for annotation and view direction.
- **2** North arrow annotation.

Example 2

The global coordinate system is selected for **Annotation Coordinate System** and a local coordinate system is selected for **View Direction Coordinate System**. North on the local coordinate system is rotated 90° counter-clockwise relative to the global coordinate system. A plan view is created, for which North of the local view direction coordinate system points up, rotating the view 90° clockwise. Because the North arrow annotation is defined by the global coordinate system, the North arrow annotation points to the right.



Orientation Properties

Orientation Rule

Specifies the rules available for orientations for the current view type. The **Orientation Rule** drives the values for the **Orientation View Direction** and **Orientation Up Direction** properties. This property is only shown for Ruleset view styles.

Orientation View Direction

Indicates the elevation direction in which you want the drawing view to "look" at the drawing objects. This property is only shown for Ruleset view styles.

Orientation Up Direction

Indicates the x- and y-axis direction with which you want the drawing view to "look" at the drawing objects. This property is only shown for Ruleset view styles.

Scale Properties

Scale Family

Specifies a scale family for the drawing view, such as Metric, Imperial, or Manual Scale. When you select a scale family, the **Scale** property values update. Selecting **Manual Scale** shows all scales, metric or imperial.

User Selected Scale

Specifies a scale to use for the drawing view. The **Scale** values update depending on the selected **Scale Family**.

■ NOTES

- For Custom scale, the default is the unit of measure setting of the document. For example, if you type values of 1 in to 1 ft, the values are converted to mm if that is the default unit of measure.
- Do not use negative values when typing custom scale values.

Navigation Rule Properties

HngSupSimpleNavigator.dll

Returns support components, supporting objects, supported objects, and design children. This rule also returns the control points on the components. This rule is specific to objects created in the Hangers and Support task.

HngSupRangeNavigator.dll

Returns support components, supporting objects, supported objects, and design children collections that extend the Hangers and Support range. This rule also returns the control points on the components. This rule is specific to objects created in the Hangers and Support task, and operates similarly to the **HngSupSimpleNavigator.dll** rule.

DrawingSpoolNavigator.dll

Returns the spool, its connected parts and their features. This rule is specific to pipe spools.

Assembly Navigator.dll

Returns the assemblies, pipe spool, penetration spool, its connected parts, and their features. This rule is specific to assemblies.

Margin Properties

Margin Left/Margin Right/Margin Top/Margin Bottom

Defines the marginal distance for the drawing view. The margin is the distance surrounding the drawing view SmartFrame. This area is used for labels and dimensions that have been designed to incorporate margins in their search for clear space.

VIEW

For example, you could have several drawing views, each with different margin settings:

Other General Properties

Look Direction

Indicates the direction in which you want the drawing view to "look" at the associated volume or objects.

Description

Describes the content of the view. This description is optional.

View Offset

Expands the view on the drawing sheet in all directions in order to prevent matchline distortion and grid plane clipping. The default offset value is **2 mm** for composed views and **0 mm** for ruleset views. The offset does not affect the size of the volume that is associated with the view.

■ NOTES

- View Offset is only available for views that are associated to a volume. Section and detail views do not have this property value.
- Negative values are not permitted for this property.
- In ruleset views, if you change the view style before editing View Offset for the first time, View Offset displays the offset value as defined in the .xml template of the view style. Make sure that your View Offset value is correct before closing the Drawing View Properties dialog box.

Flush Threshold

Sets a parameter for memory management, and helps improve drawing update performance. When the number of objects processed for a drawing document reaches the **Threshold** value, they are removed from memory. If they are needed later, they are recalled from the database. The **Flush Threshold** property is only available for composed drawing documents. The default value is **2000**, with a range of **5-5000**. Higher values are faster but use more memory, which is acceptable for smaller drawings. Lower values are slower but allow larger drawings to complete faster.

■ NOTE If a drawing document does not successfully update in the Drawings and Reports 3D task, check the error log for the drawing document. If the error log shows memory

overflow errors. lower the Flush Threshold value.

VHL Precision

Sets a parameter for Hidden Line removal processing to consider two lines as identical. This property setting has a direct impact on the visibility of the lines in the drawing. It improves drawing update performance, but it may reduce drawing quality. The **VHL precision** property is only available for composed drawing documents. The default value is **0.000001**, with a range of **0.001 to 0.000001**. The smaller the value, the more accurate the graphic elements are in the 2D view.

NOTE If some intersections of complex surfaces appear on/off along the curve, the precision of the VHL may be too restrictive compared with the precision by which the surfaces were created. Lowering the **VHL Precision** value may help the display of the intersections, but lowering the value too much could degrade the overall quality and the visibility of the drawing details.

Geometry Validation

Sets a parameter for drawing completion and quality to improve drawing update performance. The **Geometry Validation** property is available for composed drawing documents only. The default value is **Off**. When set to **Off**, the drawing document completes, but invalid geometries are left out. If set to **On**, the drawing document does not complete if invalid geometries are encountered during update.

Angle for Target Evaluation (Marine mode only)

Defines the allowable angle from the view plane for a target plate. The **Angle for Target Evaluation** property is available for the scantling view style only. Additionally, this property is used only with the **Place Detail View** or the **Place Section View** command in the SmartSketch Drawing Editor and only when a reference plane selected for the **Any system, part, or reference plane** Drawing by Rule query has been set. When plates or profiles are selected in the query, target evaluation is determined by a combination of levels and type of connection. The default value is 20°.

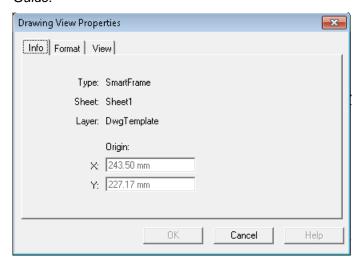
■ NOTES

- The angle is displayed in the units specified in the session file, which is degrees by default.
- For more information about Any system, part, or reference plane, see Drawings by Rule Queries in Queries Tab (Setup Dialog Box - Drawings by Rule) (on page 115).

Drawing View Properties Dialog Box (Place View Command) - Steel Order Drawings

Specifies properties for the selected drawing view in a steel order drawing. You can access this dialog box when you select and right-click on a drawing view and then select **Properties** on the shortcut menu.

For more information on creating a scantling drawing, see the *Orthographic Drawings User's Guide*.



Info Tab (Drawing View Properties Dialog Box) (on page 276)

Format Tab (Drawing View Properties Dialog Box) (on page 277)

View Tab (Drawing View Properties Dialog Box - Drawings View Explorer) (on page 283)

See Also

Place View Command (on page 275)

Info Tab (Drawing View Properties Dialog Box)

Specifies general view properties.

Type

Displays the category of the selected element.

Sheet

Displays the name of the drawing sheet that contains the selected element.

Layer

Shows the layer that contains the selected element.

Origin

Specifies the coordinates, or location, of an element along the X- and Y-axes.

Format Tab (Drawing View Properties Dialog Box)

Formats the frame around a drawing view.

Show Border

Displays the frame around the object.

Color

Sets the color of the frame.

Line Width

Sets the line thickness on the frame.

Line Type

Overrides a line type for a drawing sheet or embedded object and sets another line style for an element or a linked object.

View Tab (Drawing View Properties Dialog Box - Drawings View Explorer)

Sets the drawing view style and other properties for a selected drawing view in a steel order drawing.

Style

Specifies a view style, which includes rules for filters, updates, and graphics. The view style controls the output characteristics of the view on the generated drawing. The list displays the 10 most recently used view styles in the session. Click **More...** to display the **Select View Style** dialog box.

NOTE (Marine mode only) To display only RulesetStyles in marine mode, you need to remove the marine and material handling productIDs from the KeyPlan.xml, Orthographic.xml, and Spatial.xml files as shown in the example below. The .xml files are located in the [Reference Data Folder]\SharedContent\Drawings\Catalog\MetaStyles folder.

Original

Name

Specifies a name for the drawing view.

Coordinate System Properties

Coordinate System

Specifies a coordinate system to include in the drawing.

Orientation Properties

Orientation Rule

Specifies the rules available for orientations for the current view type. The Orientation Rule

drives the values for the **Orientation View Direction** and **Orientation Up Direction** properties. This property is only shown for rule set view styles.

Orientation View Direction

Indicates the elevation direction in which you want the drawing view to "look" at the drawing objects. This property is only shown for rule set view styles.

Orientation Up Direction

Indicates the x- and y-axis direction with which you want the drawing view to "look" at the drawing objects. This property is only shown for rule set view styles.

Scale Properties

Scale Family

Specifies a scale family for the drawing view, such as Metric, Imperial, or Manual Scale. When you select a scale family, the **Scale** property values update. Selecting **Manual Scale** shows all scales, metric or imperial.

User Selected Scale

Specifies a scale to use for the drawing view. The **Scale** values update depending on the selected **Scale Family**.

■ NOTES

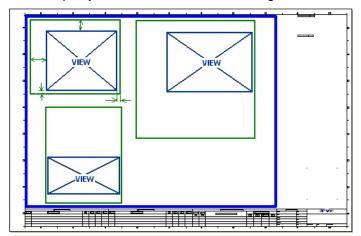
- For Custom scale, the default is the document's unit of measure setting. For example, if you type values of 1 in to 1 ft, the values are converted to mm if that is the default unit of measure.
- Do not use negative values when typing custom scale values.

Margin Properties

Margin Left/Margin Right/Margin Top/Margin Bottom

Defines the marginal distance for the drawing view. The margin is the distance surrounding the drawing view SmartFrame. This area is used for labels and dimensions that have been designed to incorporate margins in their search for clear space.

For example, you could have several drawing views, each with different margin settings:



Other General Properties

Description

Provides a description of the drawing view. This description is optional.

View Offset

Expands the view on the drawing sheet in all directions in order to prevent matchline distortion and grid plane clipping. The default offset value is **2 mm** for composed views and **0 mm** for ruleset views. The offset does not affect the size of the volume that is associated with the view.

■ NOTES

- **View Offset** is only available for views that are associated to a volume. Section and detail views do not have this property value.
- Negative values are not permitted for this property.
- In rule set views, if you change the view style before editing View Offset for the first time, View Offset displays the offset value as defined in the .xml template of the view style. Make sure that your View Offset value is correct before closing the Drawing View Properties dialog box.
- View annotations, such as the ruler and view name, are automatically adjusted in relation to the View Offset value.

VHL Precision

Sets a parameter for Hidden Line removal processing to consider two lines as identical. This property setting has a direct impact on the visibility of the lines in the drawing. It improves drawing update performance, but it may reduce drawing quality. The **VHL precision** property is only available for composed drawing documents. The default value is **0.000001**, with a range of **0.001 to 0.000001**. The smaller the number, the more accurate the graphic elements are in the 2D view.

Geometry Validation

Sets a parameter for drawing completion and quality to improve drawing update performance. The **Geometry Validation** property is available for composed drawing documents only. The default value is **Off**. When set to **Off**, the drawing document completes, but invalid geometries are left out. If set to **On**, the drawing document does not complete if invalid geometries are encountered during update.

View Cone Angle

Displays the cone angle value. This property is only available when you select the **Steel Order Extensions.xml** file as the **Additional View Inputs** property value in the **Edit Ruleset View Style** dialog box. For more information, see *General Tab (Edit Ruleset View Style Dialog Box) (Edit Ruleset View Style Dialog Box) (Edit Ruleset View Style Dialog Box)* in the *Smart 3D Drawings and Reports Reference Data Guide*.

See Also

Drawings View Explorer (on page 256)

Automatic Resize Behavior of Composed Views

For Composed drawings, views that are too small to display the associated volume or are larger than the volume are automatically resized to fit unless the view is set to **Fit to Scale** or is

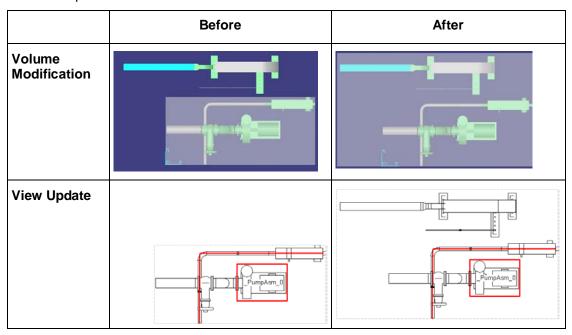
managed by a region. The view size grows or shrinks from the center of the view and view proportions may change after the resize. The drawing must be saved to make the resize of the view permanent.

NOTE You cannot crop a composed drawing view that is scaled. Views that are set to **Fit to Scale** can be cropped.

The following examples are common workflows that are affected by the automatic resizing behavior of the composed drawing view.

Overlapping Volumes Modification

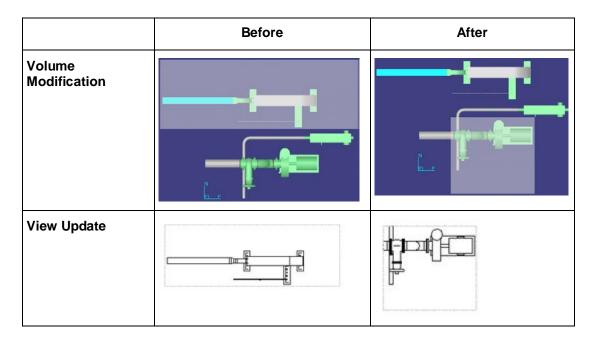
If you manually shift the sides of an associated volume such that the new volume location overlaps a portion of the old volume, the associated view resizes to mirror the volume changes. You must update the drawing or drawing view for these changes to take place. Manually-placed graphics and labels stay in the correct location on the drawing after the volume size modification and view update.



Non-overlapping Volumes Modification

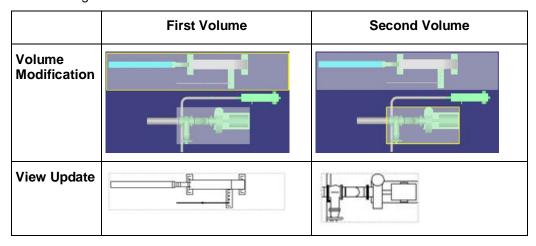
If you manually shift the sides of an associated volume, or move the volume, such that the new volume location does not overlap a portion of the old volume, the associated view resizes symmetrically around the previous center point of the view. If the volume is moved to a new location, the view does not resize. In either case, the view contents are updated with the volume contents. You must update the drawing or drawing view for these changes to take place.

Before	After



View to Volume Association

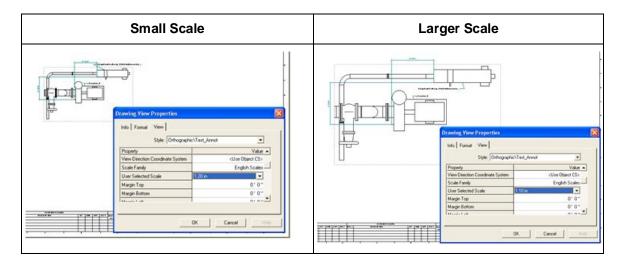
If you associate a view to a volume with the **Associate Objects to View** command, the view resizes symmetrically around the center point to match the volume size. The view resizes to match a larger or smaller volume.



View Scale Modification

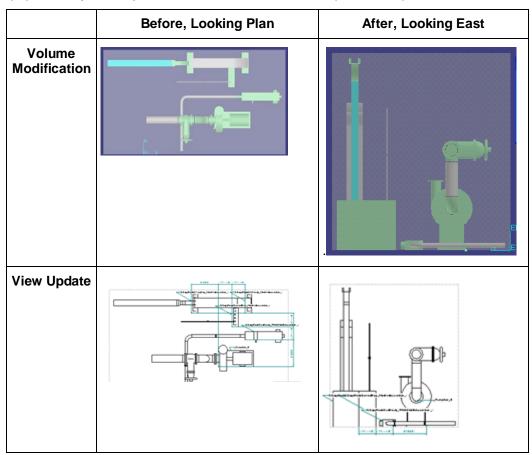
If you change the view scale on the **Drawing View Properties** dialog box, the view resizes symmetrically around the center point to accommodate the new scale. Depending on the scale change, the view grows or shrinks in size. You do not need to update the view to see these changes, but in order to populate any new objects that are inside the view, an update is required.

Small Scale	Larger Scale
-------------	--------------



View Look Direction Modification

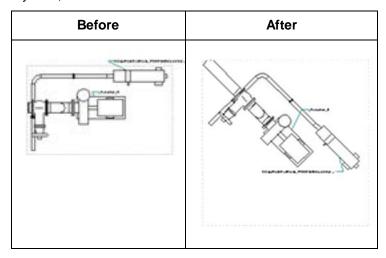
If you change the **Look Direction** from the **Drawing View Properties** dialog box, the view resizes symmetrically around the center point to match the volume size according to the new look direction. You do not need to update the view to see these changes, but in order to populate any new objects that are inside the view, an update is required.



View Direction Coordinate System Modification

If you change the **View Direction Coordinate System** in the **Drawing View Properties** dialog box, the view resizes symmetrically around the center point to accommodate the new view direction coordinate system. You do not need to update the view to see these changes, but in order to populate any new objects that are inside the view, an update is required.

NOTE If the new coordinate system has the same orientation as the previous coordinate system, the view does not resize.



Place a Manual View

The following workflows allow you to place a manual view in a Ruleset drawing either by Parts or by Reference Plane and Block.

See Also

Place a Manual View (By Parts) (on page 289)
Place a Manual View (By Reference Plane and Block) (on page 290)

Place a Manual View (By Parts)

In this workflow, you directly select detailed parts to associate with the view.

- 1. In Smart 3D, switch to a task other than Drawings and Reports, such as Molded Forms or Structural Detailing.
- Select Tools > Drawing Console.

The **Drawing Console** dialog box displays.

3. Right click a drawing document and select Edit.

SmartSketch Drawing Editor displays. In the Drawings View Explorer, a single sheet displays as a child of the drawing document.

4. In SmartSketch Drawing Editor, click the **Place View** accommand. Click and drag in the drawing area to place a graphic view.

The **Drawing View Properties** dialog box displays.

5. On the **View** tab, select **More** for the **Style** property.

The **Select View Style** dialog box displays.

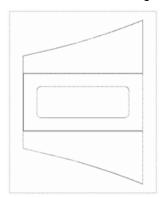
- 6. Navigate through the hierarchy to **Ruleset Styles**. Select an appropriate steel order ruleset view style, such as **Steel Order (Decks)** or **Steel Order (Generic)**, and click **OK**.
 - Additional properties display in the View tab of the Drawing View Properties dialog box.
- 7. On the View tab, type a value for Name.
- 8. On the View tab, select the needed values for Coordinate System, Scale Family, and User Selected Scale.
- 9. On the View tab, select Scantlings for Plate / Profile as the Orientation Rule value.
- 10. Click **OK**.

In the Drawings View Explorer, the view displays as a child of the drawing sheet.

- 11. Select the graphic view in the drawing area and click Associate Objects to Views 3.
- 12. Select the 3D application window to make it the active window.

The Associate ribbon displays in the 3D window.

- 13. To associate parts, select **1. Structural Parts or Plane** as the value for **Query** on the **Associate** ribbon.
- 14. In the Workspace Explorer, select plates to associate to the view. Select detailed parts, light (non-detailed) parts, systems, or leaf systems.
- 15. Click **Finish** on the **Associate** ribbon to complete the association to the drawing view. In SmartSketch Drawing Editor, preview geometry of the selected plate parts displays.



Place a Manual View (By Reference Plane and Block)

In this workflow, you select a grid plane or an offset from a grid plane to associate parts with the view and select a block, assembly block or assembly to clip the plane selected. You can use the workflow to place a major view, such as the view for a deck, transverse bulkhead, or longitudinal bulkhead.

- 1. In Smart 3D, switch to a task other than Drawings and Reports, such as Molded Forms or Structural Detailing.
- 2. Select Tools > Drawing Console.

The **Drawing Console** dialog box displays.

3. Right click a drawing document and select Edit.

SmartSketch Drawing Editor displays. In the Drawings View Explorer, a single sheet displays as a child of the drawing document.

4. In SmartSketch Drawing Editor, click the **Place View** accommand. Click and drag in the drawing area to place a graphic view.

The **Drawing View Properties** dialog box displays.

5. On the **View** tab, select **More** for the **Style** property.

The Select View Style dialog box displays.

6. Navigate through the hierarchy to **Ruleset Styles**. Select an appropriate steel order ruleset view style, such as **Steel Order (Reference Plane)** or **Steel Order - Reference Plane** (**Generic**), and click **OK**.

Additional properties display in the View tab of the Drawing View Properties dialog box.

- 7. On the View tab, type a value for Name.
- 8. On the View tab, select the needed values for Coordinate System, Scale Family, and User Selected Scale.
- 9. On the View tab, select Scantlings for Plate / Profile as the Orientation Rule value.
- 10. Click **OK**.

In the Drawings View Explorer, the view displays as a child of the drawing sheet.

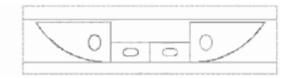
- 11. Select the graphic view in the drawing area and click Associate Objects to Views 3.
- 12. Select the 3D application window to make it the active window.

The Associate ribbon displays in the 3D window.

- 13. To associate parts by grid plane, select 1. Structural Parts or Plane as the value for Query on the Associate ribbon. In the Workspace Explorer, select a reference plane coincident with the needed plate parts. For the example in this workflow, a transverse reference plane is selected.
- 14. To associate parts by an offset from a grid plane, select 7. Offset (Optional) as the value for Query on the Associate ribbon. In the Workspace Explorer, select a reference plane. Type an offset value in Value on the Associate ribbon.
- 15. Select plate parts to associate to the view. For the example in this workflow, select plate parts.
- 16. Select **2. Block, Volume or Assembly (Optional)** as the value for **Query** on the **Associate** ribbon.
- 17. In the Workspace Explorer, select the **Assembly** tab.
- 18. Select a block, assembly block or assembly that will define the extents of the plane that is associated with the view being created.
- 19. Click Finish on the Associate ribbon to complete the association to the drawing view.

In SmartSketch Drawing Editor, preview geometry of the plate parts displays. Only parts within the selected block or assembly and coincident with the selected reference plane are

associated with the view. The block boundaries are also displayed.



Place a Manual View for Non-Shell Plates

In this workflow, you use additional options in the **Associate** ribbon to place a manual view for non-shell plates. These options display when you select the **Use Expansion** query in the **Query** tab in the **Edit Ruleset View Style** dialog box. For more information, see Edit Ruleset View Style Dialog Box.

- 1. In Smart 3D, switch to a task other than Drawings and Reports, such as Molded Forms or Structural Detailing.
- 2. Select **Tools** > **Drawing Console**.

The **Drawing Console** dialog box displays.

3. Right click a drawing document, and select Edit.

SmartSketch Drawing Editor displays. In the Drawings View Explorer, a single sheet displays as a child of the drawing document.

4. In SmartSketch Drawing Editor, click the **Place View** accommand. Click and drag in the drawing area to place a graphic view.

The **Drawing View Properties** dialog box displays.

5. On the **View** tab, select **More** for the **Style** property.

The Select View Style dialog box displays.

6. Navigate through the hierarchy to **Ruleset Styles**. Select an appropriate steel order ruleset view style, such as **Steel Order (Reference Plane)**, and click **OK**.

Additional properties display in the View tab of the Drawing View Properties dialog box.

- 7. On the View tab, type a value for Name.
- 8. On the View tab, select the needed values for Coordinate System, Scale Family, and User Selected Scale.
- 9. On the View tab, select Shell Expansion for Plate as the Orientation Rule value.
- 10. Click **OK**.

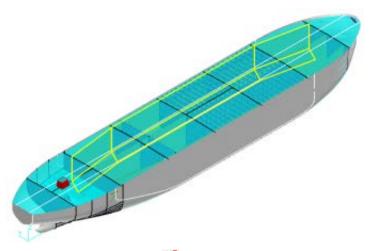
In the Drawings View Explorer, the view displays as a child of the drawing sheet.

- 12. Select the 3D application window to make it the active window.

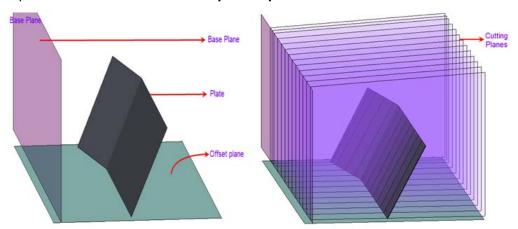
The **Associate** ribbon displays in the 3D window.

13. Click **Expansion Surface** \$\infty\$, and then select the root or leaf plate systems to see in the manual view.

NOTE You can only select systems, not parts.



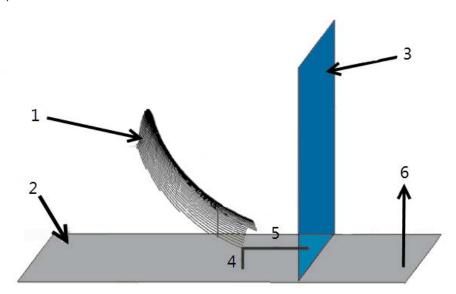
14. Click **Cutting Plane Axis** (1), and then select the axis whose child planes are parallel to the expansion direction from the **Workspace Explorer**.



NOTE Girth lengths are measured along curves defined by the intersection of each plate system and each cutting plane.

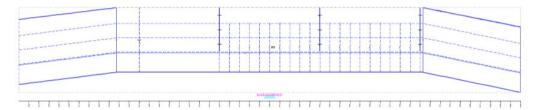
15. Click **Base Plane Definition** , and then select the base plane that defines the start point for girth measurements.

16. Specify any additional options as necessary. For example, you can select the offset plane, as shown below. For more information, see *Associate Objects to View Command* (on page 300).



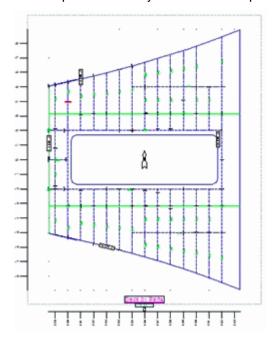
- 1 Plate Curve
- 2 Offset Plane
- 3 Base Plane
- 4 Distance from the offset plane to the start point of the plate curve
- 5 Distance from the base plane to the start point of the plate curve
- 6 Normal of the offset plane pointing to the plate
- 17. Click **Finish** on the **Associate** ribbon to complete the association to the drawing view.

 In SmartSketch Drawing Editor, preview geometry of the selected non-shell plates displays.



Update a Manual View

- 1. Right-click the out-of-date view in the Drawings View Explorer and select **Update**.
- 2. After the view updates, the view frame in the graphics area shows all geometry defined by the component view style. The view is up-to-date in the Drawing View Explorer.



■ NOTES

- You can delete a manual view permanently, or select UnAssign to move the view in the UnAssigned Folder. If you delete a view that is a parent of other views, such as a detail or section view, the Convert or Delete dialog box displays. Select Convert to independent drawing view(s) to save the child view as an independent drawing view, or select Delete to delete the child view.
- A sheet cannot be deleted unless all views on the sheet are removed first.
- All views within the drawing must be up-to-date or the drawing status will be out-of-date.
- For information on the 2D commands available for editing, see the *SmartSketch Drawing Editor Help*.

Move a Drawing View

Plant, marine, and material handling mode drawing views can only be moved while holding the **ALT** key when selecting the view. This includes drawing, section, key plan, and detail views that contain any graphics.

- 1. Press and hold the **ALT** key.
- 2. Select a view.

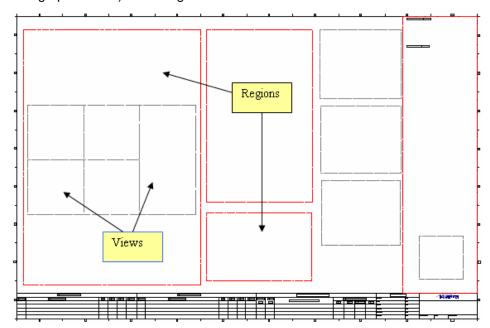
3. Drag the view to a new location.

■ NOTES

- Drawing views that have not been updated can be moved without pressing the ALT key.
- Empty views and embedded reports views can be moved without pressing the ALT key.
- The Move, Nudge, Drag, and Select All commands found on the Change toolbar also requires the use of the ALT key when selecting a view.
- You must use the Fence Select or Select All commands to select multiple views. You cannot use the CTRL key to select multiple views.

Place Region Command

Creates a new drawing region in the drawing area. A *region* is a container that controls the arrangement of the views inside by means of an associated layout style. Views controlled by a region are called *managed* views. You can manage any type of drawing view (report, key plan and graphic views) with a region.



This command is available in **SmartSketch Drawing Editor** when you edit a layout template or edit an existing drawing.

When you click **Place Region**, you click and drag to define the new region. The **Region Properties** dialog box appears so that you can define the properties associated with the region.

After creating the region and defining its properties, use the **Place View** or **Place Report View** command to place views within the region.

Region Behavior

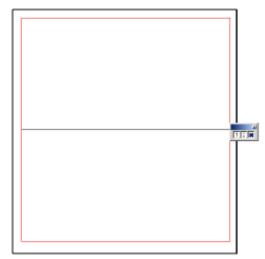
You can place a drawing view outside a region, but this makes the drawing view an "unmanaged view," meaning the properties of the region do not control the drawing view. However, if a view is inside a region or touching a region, the region manages the drawing view and, when you update the drawing in the 3D Drawings and Reports task, the software pulls the drawing view into the region and updates it based on the region layout style and layout processor.

- If a region refuses to accept a drawing view (for example: the region is defined for three views, and you are attempting to add a fourth view), the drawing view is added as an unmanaged view, just outside and to the upper left of the region. If another region occupies this space in the drawing area, the unmanaged view is placed as close to the original region as possible.
- If a drawing view "straddles" two or more regions, the region that contains more of the drawing view manages it. If the drawing view equally straddles two or more regions, the software uses the first drawing view point to measure distance and determine which region manages the drawing view.
- If a region contains a drawing view and the drawing view properties make it ineligible for the region, the software removes the drawing view from the region automatically and places it in the upper left of the drawing area, outside of other regions.
- If you switch the border of a drawing using the Switch Border command, views may reposition or resize with the new border template based on the following conditions:
 - Managed views are resized and repositioned according to the new border template.
 - Unmanaged views are not resized and repositioned.
 For more information, see Switch Border (on page 210).

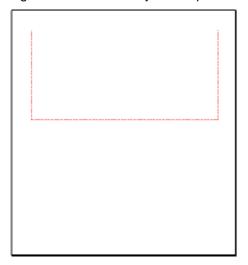
Deleting a Region

Regions are used to manage the positioning of views within a layout template. When you delete a region, any views that were managed by the region become "unmanaged" views.

1. To delete a region from a layout template, you should use **PickQuick** to select the outer boundary of the region.



After selecting the region outer boundary, select Edit > Delete to remove the region
definition. You can also press the Delete key on the keyboard. Any views placed within the
region remain on the layout template as unmanaged views.



See Also

Region Properties Dialog Box (on page 298)

Region Properties Dialog Box

Specifies properties for the selected region. This dialog box displays after you have created a new region using the **Place Region** command, which is only available in **SmartSketch Drawing Editor**.

Description

Provides a description of the region.

Layout Style

Specifies the layout style to associate with this region. The layout style dictates the order and placement of the drawing views managed by the region. Select **More** to display the **Define Layout Style** dialog box and select the layout style from a list of available styles. For more information, see *Define Layout Style Dialog Box* (on page 299).

Maintain Aspect

Specifies whether or not the aspect ratio for each drawing view and region is maintained within the layout. This property is useful when you are "stretching" a layout into a border or when you are changing the border.

Lock Region

Locks the positioning of the drawing views within the region. This property also prevents you from adding or deleting existing views.

See Also

Convert Excel Spreadsheet Reports to Native Text Box Format Custom Command (on page 403)

Place Region Command (on page 296)

Define Layout Style Dialog Box

Specifies a layout style to associate with a drawing region. This dialog box displays when you select **More** in the **Value** field for the **Layout Style** property on the **Region Properties** dialog box. The hierarchy shows all available layout styles.

See Also

Convert Excel Spreadsheet Reports to Native Text Box Format Custom Command (on page 403)

Place Region Command (on page 296)

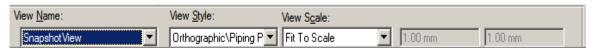
Region Properties Dialog Box (on page 298)

Place Snapshot View Command

Creates a new snapshot drawing view in the drawing area. This command is only available in **SmartSketch Drawing Editor** when you create a new drawing or open an existing drawing from a 3D task.

■ NOTE Before you place snapshot views, you need to create a snapshot using the 3D task Tools > Snapshot View command. If you do not have any snapshot views, a message displays and you can go back to a 3D task and create the snapshot views.

When you click **Place Snapshot View** w, you click and drag to define the new view. You specify the snapshot view properties in the **Snapshot View** ribbon displayed above the drawing area.



When you edit a drawing that contains a snapshot view, you can right-click the view and select **Properties** to change the style or scale used in the snapshot view.

To update the contents of the snapshot view, right-click the view and select **Update View**.

See Also

Update View Command (on page 399)

Place Snapshot View Ribbon (on page 299)

Drawing View Properties Dialog Box (Drawing View Shortcut Menu) (on page 382)

Place Snapshot View Ribbon

Sets options for placing a snapshot drawing view. These options include the view name, style, and scale.

View Name

Displays the name of the view you created in a 3D task.

View Style

Specifies a view style, which includes rules for filters, updates, and graphics. The view style controls the output characteristics of the view on the generated drawing. The list displays the 10 most recently used view styles in the session. Click **More...** to display the **Select View Style** dialog box.

View Scale

Specifies a scale. You can choose **No Scale** if the view is not to scale. Choose **Custom** if you want to specify your own scale and then type values in the two boxes at the right of the ribbon. If you choose **Custom** for the scale, you must type values that are greater than zero in the boxes at the right. If you select **Fit to Scale**, the software places the snapshot to fit the scale of the view you place. If you select one of the scale families, such as **Metric Scales**, the software sets the cursor for placement at the selected scale.

See Also

Place Snapshot View Command (on page 299)

Associate Objects to View Command

Allows you to interactively specify volumes or other model objects as input for the selected drawing views. If the view is a key plan or report view, you specify an existing drawing view. This command is also used in creating composed drawings in a 3D task. The command is only available in **SmartSketch Drawing Editor** when you create a new drawing or open an existing drawing from a 3D task.

When you select a view and click **Associate Objects to View** the **Associate Objects to View** ribbon appears on the 3D task window. **Associate Objects to View** turns off **2D/3D Selection** so that you can move back and forth between the 3D application and SmartSketch Drawing Editor, selecting views and associating volumes or model objects.

Orthographic View Ribbon

If you are associating volumes to a graphic drawing view, the following ribbon options appear:

Select View

Shows either all of the graphic views in the drawing or a list of selected views from the drawing. You can multi-select views before running the **Associate Objects to View** command to populate the list with only the selected views. As you click volumes to associate them to the currently shown view, the list scrolls to the next view automatically. The views are listed in alphabetical order.

Exclude Objects

Allows you to select objects to exclude from the drawing view or to view objects that have been previously selected for exclusion from the drawing view. When you click **Exclude Objects**, the **Accept/Cancel** buttons enable on the ribbon so that you can confirm the selection. Excluded objects are highlighted.

■ NOTES

- You should associate your volume before excluding objects from the volume.
- You can select the objects from the model, or from the Workspace Explorer, but you cannot select referenced objects that originate from files on the Reference tab of the Workspace Explorer.
- You can clear objects that have previously been excluded from the drawing view by clicking on the object.
- If you exclude an object while the related tab of the **Workspace Explorer** is active, the child objects are automatically excluded. For example, if the **System** tab is active when you exclude a system object, then all of the children objects are also excluded. If the

related WSE tab is not active, only the object that you specifically select is excluded. For example, if you exclude a pipe run while the **Space** tab is active, the child parts of the pipe run are not excluded.

Select Volume

Sets the command in select mode so that you can select a volume graphically or from the **Workspace Explorer**. After you select a volume in the workspace window, you can use the other controls on the ribbon to further define the objects collected for the drawing view. This is the default setting when the command ribbon appears.

× Cancel

Clears all objects selected for exclusion from the drawing view and returns you to the previous **Select volume** command step.

Accept

Accepts all objects selected for exclusion from the drawing view and returns you to the **Select volume** command step.

Filter

Specifies a filter to restrict the objects to include in the volume. You can right-click and select **More** to select a filter from an existing list of filters or to create a new filter.

■ NOTES

- You should not use filters from the My Filters folder when using the Filter option on the Associate Objects to View ribbon.
- Section and detail views inherit filters used by the Filter option from the parent view.

Properties

Displays the properties for the selected filter.

🗞 Clear Filter

Clears the currently selected filter. This button only enables when you have selected a filter in the **Filter** dropdown field.

Look Direction

Indicates the view direction for the drawing view. The current setting is taken from the drawing view properties, set when the view was created with the **Place View** command in SmartSketch Drawing Editor. If you change the **Look Direction**, you are also changing the view direction property on the drawing view. If you select **From Active View**, the software uses the **Look Direction** of the active graphic view.

Rule Set View Ribbon (List-Based)

When the graphic view has a rule set view style, the following ribbon options appear:

Query

Specifies the query to apply to the drawing view. The selected query tells the software which objects to include in the drawing view.

▶ NOTE A view query specifies the type of model object or parameter value used by the view definition. The list of available queries is specific to the selected package. If you are a reference data administrator customizing a drawing package, view queries are parts of the

rule set defined on the **Actions** tab of the **Edit Ruleset View Style** dialog box in the Drawings and Reports task. For more information, see the appropriate drawing type under *Drawings by Rule View Style Rule Sets* in the *Drawings and Reports Reference Data Guide*.

Value

Specifies a value required by the selected query. The type of value is defined by the query. You may need to select a value from a list, type a value, or turn an option on or off.

Finish

Saves the specified query and value to the selected drawing view.

Rule Set View Ribbon (Step-Based)

When the graphic view has a rule set view style, the following ribbon options appear:

选 Associate Objects to View

Displays the default parts available for the view. Select the needed parts to add them to the view.

- If all needed parts are displayed, select them in the graphic view, and then click Finish to return to SmartSketch Drawing Editor.
- If all needed parts are not displayed, select them using the **Create Volume** 🗗 or **Gather Objects** 🖫 options.

ℰ Create Volume

Changes the default volume definition for the drawing view. For more information, see *Create Volume Options*, below.

Gather Objects

Changes the default part type definitions to include in the volume. For more information, see *Gather Objects Options*, below.

Orient View

Changes the default view direction and orientation. For more information, see *Orient View Options*, below.

TIP With this option, you can orient the drawing view by the local coordinate system of a part selected in the graphic view.

Finish

Saves the specified options to the drawing view and returns to SmartSketch Drawing Editor.

Reject

Rejects the selected options.

Accept

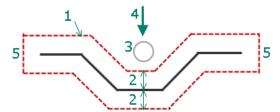
Accepts the selected options.

Create Volume Options

Volume Creation Rule - Defines the volume for the view by one of the following methods:

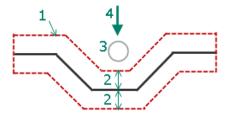
Create volume along surface of input parts with boundary extents - The volume follows the surfaces of the selected parts resulting from Any system, part, or reference plane. The volume depth (normal to the view) is extended by the values of Volume growth into drawing and Volume growth out of drawing. The volume area (in the plane of the view) is extended to (or clipped by) the boundaries defined by Boundary (Block, volume, or assembly).

Example: Corrugated bulkhead



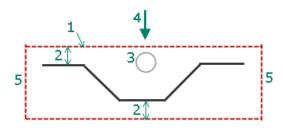
- 1 Cross-section of volume
- 2 Volume growth
- 3 Additional part not gathered into view
- 4 View direction
- 5 Volume extended to boundaries
- Create volume along surface of input parts without boundary extents The volume follows the surfaces of the selected parts resulting from Any system, part, or reference plane. The volume depth (normal to the view) is extended by the values of Volume growth into drawing and Volume growth out of drawing. The volume area (in the plane of the view) is not extended to (nor clipped by) the boundaries defined by Boundary (Block, volume, or assembly).

Example: Corrugated bulkhead



- 1 Cross-section of volume
- 2 Volume growth
- 3 Additional part not gathered into view
- 4 View direction
- Create volume from range box around input parts with boundary extents The volume is a rectangular range box around the selected parts resulting from Any system, part, or reference plane. The volume depth (normal to the view) is extended by the values of Volume growth into drawing and Volume growth out of drawing. The volume is extended to (or clipped by) the boundaries defined by Boundary (Block, volume, or assembly). See the corrugated bulkhead example below.

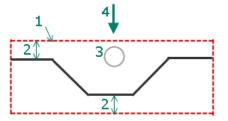
Example: Corrugated bulkhead



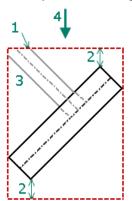
- 1 Cross-section of volume
- 2 Volume growth
- 3 Additional part gathered into view
- 4 View direction
- **5** Volume extended to boundaries

The volume is a rectangular range box around the selected parts resulting from Any system, part, or reference plane. The volume depth (normal to the view) is extended by the values of Volume growth into drawing and Volume growth out of drawing. The volume is not extended to (nor clipped by) the boundaries defined by Boundary (Block, volume, or assembly). See the corrugated bulkhead example below.

Example: Corrugated bulkhead



Example: Non-orthogonal member



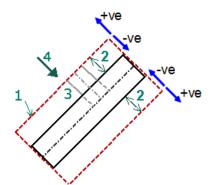
- 1 Cross-section of volume
- 2 Volume growth
- 3 Additional part gathered into view
- 4 View direction

Create oriented volume around input parts without boundary extents - The volume is a rectangular box around the parts. The box is oriented around parts to minimize the volume. The parts are typically not orthogonal, but the view direction is typically normal to the box. The volume depth (normal to the view) is extended by the values of Volume growth into drawing and Volume growth out of drawing.

■ NOTES

- If the view direction is not normal to the box, then the volume is extended to (or clipped by) the boundaries defined by Boundary (Block, volume, or assembly) instead of Volume growth into drawing and Volume growth out of drawing.
- If the view direction is normal to the box, then the volume is extended to (or clipped by) **Volume growth into drawing** and **Volume growth out of drawing**. These values can be positive or negative.

Example: Non-orthogonal member



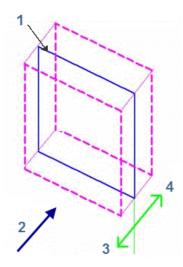
- 1 Cross-section of volume
- 2 Volume growth
- 3 Additional part gathered into view
- 4 View direction
- **Do not create volume** A volume is not used to restrict gathering using gathering rule. For example, a shell longitudinal profile view does not use a volume. It instead uses levels to gather secondary parts.

Depth Into

Defines a distance into the drawing to extend the depth of a thin volume. Parts within the thin volume are displayed in the view. A value for this query is optional.

Depth Out Of

Defines a distance out of the drawing to extend the depth of a thin volume. Parts within the thin volume are displayed in the view. A value for this query is optional.



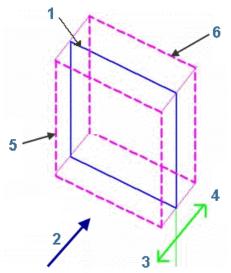
- 1 View plane
- 2 View direction
- 3 Out of drawing
- 4 Into drawing

Boundary Overlap

Defines the distance that parts extend past the view boundary.

Offset

Defines the distance from a reference plane used to create the view plane. A positive value offsets the view plane out of the drawing. A negative value offsets the view plane into the drawing. This value can be used in combination with **Volume growth into drawing** and **Volume growth out of drawing** to create a thin volume. A value for this query is optional.



- 1 Reference plane
- 2 View direction
- 3 Offset out of drawing (positive value)
- 4 Offset into drawing (negative value)
- 5 View plane offset out of drawing
- 6 View plane offset into drawing

Clip to Volume

Clips part visibility in the graphic view to the boundaries of the volume.

Gather Objects Options

Gathering Rule

Defines the type of parts gathered in the drawing view. The rule provides a definition of the objects to be gathered and drawn in a view. Select a rule that supports the type of drawing view you want to create, such as **Steel Order**, **Piping**, **Pipe Supports**, or **Assembly**.

Include all objects in volume

Defines how gathered parts are included in the drawing view. Select to gather all parts in the volume even if they are not connected to the selected parts. Clear to gather only the selected parts and parts connected to the selected parts.

Show Objects Gathered

Highlights the gathered parts.

Orient View Options

Orientation Rule

Specifies the rule used to define the view orientation. Select options such as Global Coordinate System, Local Coordinate System, Scantlings for Plate/Profile, and Scantling By Plate Normal.

View Direction

Defines the direction of the view. The available options are defined by the selected orientation rule.

Up Direction

Defines the up direction of the view. The available options are defined by the selected orientation rule.

Orient Display to View Direction

Changes the orientation of the graphic view to match the view orientation. This option is only available when **Local Coordinate System** is selected for **Orientation Rule**.

For more information on view orientation options, see *Orientation Rules* (on page 110) in the *Smart 3D Orthographic Drawings User's Guide*.

Key Plan or Report View Ribbon

When the view you selected is specified as a report or key plan view style, you need to associate a drawing view to populate the contents of the report or key plan. The following ribbon options appear:

View

Displays a list of views available for association with the report or key plan.

Finish

Returns you to SmartSketch Drawing Editor with the selected view associated to the report or key plan view.

Generic View Ribbon

When the selected graphic view is specified as a generic rule set view style, the following ribbon options display:

Associate Objects to View

Displays the default parts available for the view. Select the needed parts to add them to the view.

- If all needed parts are displayed, select them in the graphic view, and then click **Finish** to return to SmartSketch Drawing Editor.

Changes the default volume definition for the drawing view. For more information, see *Create Volume Options*, below.

Gather Objects

Changes the default part type definitions to include in the volume. For more information, see *Gather Objects Options*, below.

Orient View

Changes the default view direction and orientation. For more information, see *Orient View Options*, below.

TIP With this option, you can orient the drawing view by the local coordinate system of a part selected in the graphic view.

Finish

Saves the specified options to the drawing view and returns to SmartSketch Drawing Editor.

Reject

Rejects the selected options.

Accept

Accepts the selected options.

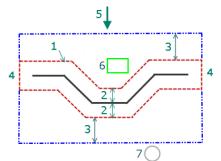
Create Volume Options

Volume Creation Rule

Defines the volume for the view by one of the following methods:

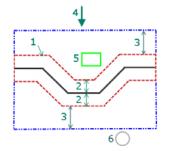
Create volume along surface of input parts with boundary extents - The volume follows the surfaces of the selected parts resulting from Any system, part, or reference plane. The volume depth (normal to the view) is extended by the values of Volume growth into drawing and Volume growth out of drawing. The volume area (in the plane of the view) is extended to (or clipped by) the boundaries defined by Boundary (Block, volume, or assembly). The gathering range is extended by the values defined by Extend Into and Extend Out Of.

Example: Corrugated bulkhead



- 1 Cross-section of volume
- 2 Volume growth
- 3 Extended gathering range
- **4** Volume extended to boundaries
- 5 View direction
- **6** Additional part gathered into view
- **7** Additional part not gathered into view
- Create volume along surface of input parts without boundary extents The volume follows the surfaces of the selected parts resulting from Any system, part, or reference plane. The volume depth (normal to the view) is extended by the values of Volume growth into drawing and Volume growth out of drawing. The volume area (in the plane of the view) is not extended to (nor clipped by) the boundaries defined by Boundary (Block, volume, or assembly). The gathering range is extended by the values defined by Extend Into and Extend Out Of.

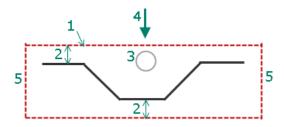
Example: Corrugated bulkhead



- 1 Cross-section of volume
- 2 Volume growth
- 3 Extended gathering range
- 4 View direction
- 5 Additional part gathered into view
- **6** Additional part not gathered into view

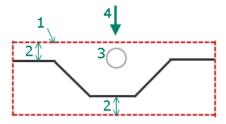
Create volume from range box around input parts with boundary extents - The volume is a rectangular range box around the selected parts resulting from Any system, part, or reference plane. The volume depth (normal to the view) is extended by the values of Volume growth into drawing and Volume growth out of drawing. The volume is extended to (or clipped by) the boundaries defined by Boundary (Block, volume, or assembly). See the corrugated bulkhead example below.

Example: Corrugated bulkhead

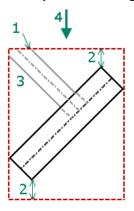


- 1 Cross-section of volume
- 2 Volume growth
- 3 Additional part gathered into view
- 4 View direction
- **5** Volume extended to boundaries
- Create volume from range box around input parts without boundary extents The volume is a rectangular range box around the selected parts resulting from Any system, part, or reference plane. The volume depth (normal to the view) is extended by the values of Volume growth into drawing and Volume growth out of drawing. The volume is not extended to (nor clipped by) the boundaries defined by Boundary (Block, volume, or assembly). See the corrugated bulkhead example below.

Example: Corrugated bulkhead



Example: Non-orthogonal member



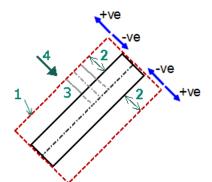
- 1 Cross-section of volume
- 2 Volume growth
- **3** Additional part gathered into view
- 4 View direction

 Create oriented volume around input parts without boundary extents - The volume is a rectangular box around the parts. The box is oriented around parts to minimize the volume. The parts are typically not orthogonal, but the view direction is typically normal to the box. The volume depth (normal to the view) is extended by the values of **Volume** growth into drawing and **Volume** growth out of drawing.

■ NOTES

- If the view direction is not normal to the box, then the volume is extended to (or clipped by) the boundaries defined by Boundary (Block, volume, or assembly) instead of Volume growth into drawing and Volume growth out of drawing.
- If the view direction is normal to the box, then the volume is extended to (or clipped by) **Volume growth into drawing** and **Volume growth out of drawing**. These values can be positive or negative.

Example: Non-orthogonal member



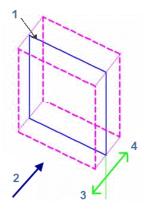
- 1 Cross-section of volume
- 2 Volume growth
- 3 Additional part gathered into view
- 4 View direction
- **Do not create volume** A volume is not used to restrict gathering using gathering rule. For example, a shell longitudinal profile view does not use a volume. It instead uses levels to gather secondary parts.

Depth Into

Defines a distance into the drawing to extend the depth of a thin volume. Parts within the thin volume are displayed in the view. A value for this query is optional.

Depth Out Of

Defines a distance out of the drawing to extend the depth of a thin volume. Parts within the thin volume are displayed in the view. A value for this query is optional.



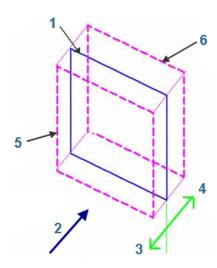
- 1 View plane
- 2 View direction
- 3 Out of drawing
- 4 Into drawing

Boundary Overlap

Defines the distance that parts extend past the view boundary.

Offset

Defines the distance from a reference plane used to create the view plane. A positive value offsets the view plane out of the drawing. A negative value offsets the view plane into the drawing. This value can be used in combination with **Volume growth into drawing** and **Volume growth out of drawing** to create a thin volume. A value for this query is optional.



- 1 Reference plane
- 2 View direction
- 3 Offset out of drawing (positive value)
- 4 Offset into drawing (negative value)
- 5 View plane offset out of drawing
- 6 View plane offset into drawing

Extend Into

Defines the distance that the gathering range box extends into the drawing. You can display parts that are outside of the thin volume and in the extended gathering range box. A value for this query is optional.

Extend Out Of

Defines the distance that the gathering range box extends out of the drawing. You can display parts that are outside of the thin volume and in the extended gathering range box. A value for this query is optional.

Olip to Volume

Clips part visibility in the graphic view to the boundaries of the volume.

Gather Objects Options

Gathering Rule

Defines the type of parts gathered in the drawing view. The rule provides a definition of the objects to be gathered and drawn in a view. Select a rule that supports the type of drawing view you want to create, such as **Steel Order**, **Piping**, **Pipe Supports**, or **Assembly**.

Include all objects in volume

Defines how gathered parts are included in the drawing view. Select to gather all parts in the volume even if they are not connected to the selected parts. Clear to gather only the selected parts and parts connected to the selected parts.

🛂 Show Objects Gathered

Highlights the gathered parts.

Orient View Options

Orientation Rule

Specifies the rule used to define the view orientation. Select options such as Global Coordinate System, Local Coordinate System, Scantlings for Plate/Profile, and Scantling By Plate Normal.

View Direction

Defines the direction of the view. The available options are defined by the selected orientation rule.

Up Direction

Defines the up direction of the view. The available options are defined by the selected orientation rule.

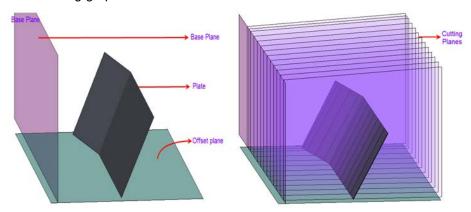
Orient Display to View Direction

Changes the orientation of the graphic view to match the view orientation. This option is only available when **Local Coordinate System** is selected for **Orientation Rule**.

For more information on view orientation options, see *Orientation Rules* (on page 110) in the *Smart 3D Orthographic Drawings User's Guide*.

Expansion View Ribbon

Provides options for generating expansion views. An overview of the required inputs is shown in the following graphic:



■ NOTE This ribbon displays only when the Associate Views to Object Command option is set to Expansion. For more information, see General Tab (Edit Ruleset View Style Dialog Box) in the Drawings and Reports Reference Data Guide. You can access this document using the Help > Printable Guides command in the Drawings and Reports task.

Expansion Surface

Select a plate system or plate part for which you want expansion drawings. You can select multiple plate systems or plate parts.

Cutting Planes Axis

Select a coordinate system axis for defining the cutting planes.

Base Plane Definition

Select a plane that is normal to the grid planes of the cutting plane axis.

Offset Plane

Select a plane that is perpendicular to the grid planes of the base plane and cutting plane axes. This plane is the reference plane from which the curve is expanded. This is an optional input.

PExpansion Direction

Defines the expansion direction. This option is available only when the base plane intersects the expansion surface. If you select **Positive Direction**, the expansion surface on the positive side of the base plane will be expanded and its corresponding drawing will be generated. If you select **Negative Direction**, the expansion surface on the negative side of the base plane will be expanded and its corresponding drawing will be generated. If you select **Both**, the expansion surface will be expanded completely and its corresponding drawing will be generated.

Block or Volume

Select a bounding block or volume. This is an optional input.

Cancel

Clears the selected options and allows you to continue the selection process.

Accept

Accepts the selected options.

Finish

Finishes the command.

For workflows using this command, see Associate Objects to Views (on page 314) and Associate Objects to a Drawings by Rule View (on page 318).

■ NOTES

- When using a Snapshot view style, all objects within the associated volume are included in the drawing view, regardless of exclusions made using Associate Objects to View
 3.
- When using a Volume view style, only the objects returned by the filters in the view style are included in the drawing view.
- An error message displays if a volume is not selected in the 3D application.
- If a 3D task process is running (for example: a recompute of relationships after moving a grid plane), an informational message displays and **Associate Objects to View** is canceled.
- Views that are too small to display the volume are automatically sized larger to fit unless the view is set to Fit to Scale or is managed by a region. Likewise, views that are larger than the volume are automatically resized to fit unless the view is set to Fit to Scale or is managed by a region. The view size grows or shrinks from the center of the view and view proportions may change after the resize. The drawing must be saved to make the resize of

the view permanent. For more information, see *Automatic Resize Behavior of Composed Views* (on page 285).

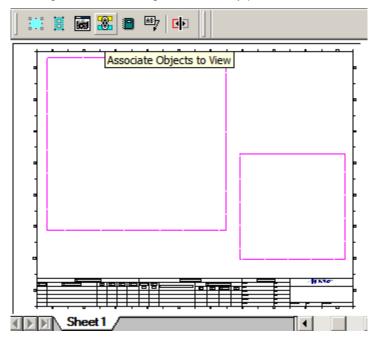
- To remove associated inputs from a view, use **Remove Associated Inputs** . For more information, see *Remove Associated Inputs Command* (on page 319).
- Section and detail views inherit filters used by the Filter option from the parent view.

See Also

Place View Command (on page 275) Update View Command (on page 399)

Associate Objects to Views

The following procedure steps you through associating volumes to drawings views in a 3D composed drawing. The procedures assumes you have already opened a 3D drawing in SmartSketch Drawing Editor to place and associate views. The example workflow uses a drawing with two drawing views already placed.

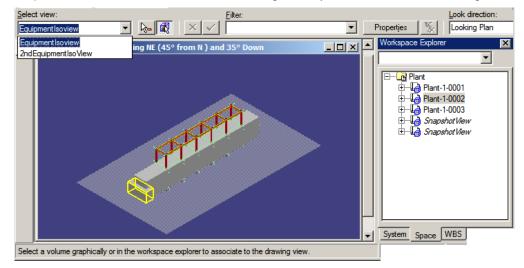


NOTE For more information on 3D composed drawings, see the *Orthographic Drawings User's Guide*. For information on the commands available for editing 3D drawings, see *Working with Drawings and Reports and SmartSketch Drawing Editor* (on page 253).

Associate Drawing Views with Orthographic View Styles

1. In SmartSketch Drawing Editor, select the orthographic drawing views that you want to associate, and then click **Associate Objects to View**.

 In the 3D application, notice the Associate Objects to View ribbon above the active window. If the selected drawing view has an orthographic view style, the Select View dropdown includes the names of the drawing views you selected in the drawing.



- Click volume graphically or in the Workspace Explorer to associate to the current drawing view. The Select View dropdown scrolls automatically to the next drawing view in the list. Click another volume to associate with the second drawing view.
- 4. When you are done associating volumes to drawing views, return to SmartSketch Drawing Editor, and click **Place View**.
- 5. To update the view contents with the associated volumes, select and right-click the view and select **Update View** on the shortcut menu. For more information, see *Update View Command* (on page 399).
- 6. Save the drawing document and exit SmartSketch Drawing Editor.
- 7. Update and complete your drawing document.

Exclude Objects from the Volume

The **Associate Objects to View** command ribbon allows you to further define the objects associated to the drawing view. You can use filters to restrict the objects collected from the volume. You can also exclude objects from the volume if you do not want them included in the drawing.

Select a volume graphically or in the workspace explorer to associate to the drawing least to

In the following graphic, the **Associate Objects to View** ribbon shows how you can use a filter, such as **Equipment**, to define the objects shown in the drawing.

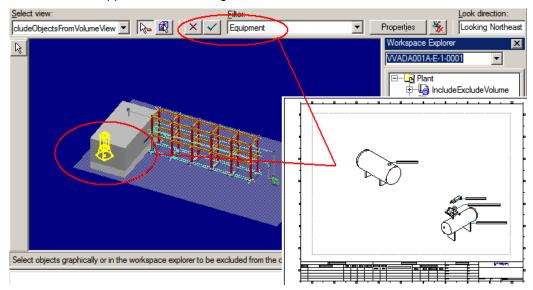
In the next example, the same **Equipment** filter is applied to the volume. However, a **Vertical Tank** is excluded from the filter. You achieve this output as follows:

- 1. Select the Filter, then click Exclude Objects ...
- 2. Select the objects inside the volume that you want to exclude from the drawing.

■ NOTES

- We recommend that you associate your volume before excluding objects from the volume.
- You can select the objects from the model, or from the Workspace Explorer, but you cannot select referenced objects that originate from files on the Reference tab of the Workspace Explorer.
- You can clear objects that have previously been excluded from the drawing view by clicking on the object.
- If you exclude an object while the related tab of the Workspace Explorer is active, the child objects are automatically excluded. For example, if the System tab is active when you exclude a system object, then all of the children objects are also excluded. If the related WSE tab is not active, only the object that you specifically select is excluded. For example, if you exclude a pipe run while the Space tab is active, the child parts of the pipe run are not excluded.
- 3. Click **Accept** \checkmark to save the change to the volume definition.

 Go back to SmartSketch Drawing Editor and update the view contents with the associated volumes with the **Update View** command on the shortcut menu of the view. The **Vertical Tank** does not appear in the drawing view.



- 5. Save the drawing document and exit SmartSketch Drawing Editor.
- 6. Update and complete your drawing document.

Associate Key Plan or Report Views

- 1. Select a key plan or report view in the drawing document.
- 2. Click Associate Objects to View 38.
- In the 3D application, select a view from the View dropdown. The View dropdown contains a list of all the drawing views available in the current drawing document open in SmartSketch Drawing Editor.
- 4. Click **Finish** to save the association to the key plan or report view.
- 5. Go back to SmartSketch Drawing Editor to update the view contents. Select and right- click the view and select **Update View** on the shortcut menu. For more information, see *Update View Command* (on page 399).
- 6. Save the drawing document and exit SmartSketch Drawing Editor.
- 7. Update and complete your drawing document.

Associate to a Drawing View with a Ruleset View Style

Ruleset view styles are only used in Smart 3D drawings.

- 1. Select a drawing view with a ruleset view style then click Associate Objects to View 3.
- In the 3D application, select a query from the Query dropdown. Depending on the query, you may need to specify a value or select objects graphically.
- 3. Click Finish to save the association to the drawing view.

- 4. Go back to SmartSketch Drawing Editor to update the view contents. Select and right- click the view and select **Update View** on the shortcut menu. For more information, see *Update View Command* (on page 399).
- 5. Save the drawing document and exit SmartSketch Drawing Editor.
- 6. Update and complete your drawing document.

See Also

Place View Command (on page 275)
Associate Objects to View Command (on page 300)

Associate Objects to a Drawings by Rule View

- 1. In Smart 3D, switch to a task other than Drawings and Reports, such as Molded Forms or Structural Detailing.
- 2. Click Tools > Drawing Console.

The **Drawing Console** dialog box displays.

- 3. Right-click an existing Drawings by Ruleset drawing document and click Edit.
 - SmartSketch Drawing Editor displays.
- - The Drawing View Properties Dialog Box displays.
- 5. On the **View** tab, select **More** for the **Style** property.
 - The **Select View Style** dialog box displays.
- 6. Navigate through the hierarchy and select a drawing view style, and then click OK.
 - Additional properties display on the View tab of the Drawing View Properties dialog box.
 - **NOTE** For more information about ruleset view style definitions, see *Define View Style Command* in the *Drawings and Reports Reference Data Guide*. You can access this document using the **Help > Printable Guides** command in the Drawings and Reports task.
- 7. On the **View** tab, type a value for **Name**, and then select the needed values for **Coordinate System**, **Scale Family**, and **User Selected Scale**.
- 8. Click OK.
- 9. Select the graphic view in the drawing area and click Associate Objects to View &.
- 10. Select the 3D application window to make it the active window.
- 11. Use the ribbon that displays in the 3D window to define the necessary options.

■ NOTES

- For more information, see *Place View Command* (on page 275) and *Associate Objects to View Command* (on page 300).
- The software displays either the **Drawing Ruleset Ribbon** or the **Expansion Ribbon**, depending on how you defined the **Associate Objects to View** options for the selected view style. For more information about this option, see *General Tab (Edit Ruleset View Style Dialog Box)* in the *Drawings and Reports Reference Data Guide*.

Remove Associated Inputs Command

Allows you to remove the associated volumes, filters, views, and other inputs from the selected view. It works with all graphic and non-graphic (report and keyplan) views. The command is available on the Compose toolbar when creating a new 3D composed drawing or editing a 3D composed drawing.

When you click **Remove Associated Inputs** a status bar message instructs you to select a single drawing view. After you select a drawing view, a confirmation message displays. Click **Yes** to continue and remove the associated inputs from the selected view, or click **No** to cancel the command.

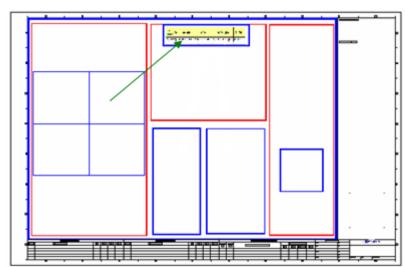
■ NOTE You cannot **Undo** or **Redo** actions performed by the **Remove Associated Inputs** command.

See Also

Place View Command (on page 275) Associate Objects to View Command (on page 300) Update View Command (on page 399)

Place Report View Command

Adds a report "view" in the drawing. The report view becomes an embedded report based on the contents of a drawing view. You could create Bill of Material, Issue Record, or a Revision report tied to the contents of a specified drawing view. This command is only available in **SmartSketch Drawing Editor** when you create a new drawing or open an existing drawing from a 3D task.



When you click **Place Report**, you click and drag to define the new report view. The **Report Properties** dialog box appears so you can define the properties associated with the report view.

When you select a report view and click **Associate Objects to View** the **Associate Objects to View** ribbon appears on the 3D task window. For more information, see *Associate Objects to View Command* (on page 300).

For information on creating new views, see Place View Command (on page 275).

For a list of reports that can be embedded in a drawing successfully, see *Reports Compatible* with *Drawings* (on page 321).

■ NOTES

- You can associate multiple reports to the same view, but you cannot associate multiple views to the same report.
- You cannot multi-select report views to associate to a single view. The relationship is one-to-one from report view to graphic drawing view. Also, you cannot multi-select report views and graphic views. Other objects (such as lines or symbols) are ignored if they are in the select set with the report view when you run the Associate Objects to View command.

See Also

Report Properties Dialog Box (Place Report View Command) (on page 320) Update View Command (on page 399) Report Properties Dialog Box (Place Report View Command) (on page 320)

Report Properties Dialog Box (Place Report View Command)

Specifies properties for the selected report view. This dialog box displays after you have placed a new report view using the **Place Report View** command or when you edit properties for an existing report view.

Name

Specifies a name for the report view.

Description

Provides a description of the report view.

Report Template

Specifies the report template to use when populating the report view. Select **More** in the **Value** list to display the **Select Template** dialog box. For more information, see *Select Template Dialog Box* (on page 321).

Report Template Output

Specifies how the report output will be formatted. Select **Excel** to output the report as an Excel spreadsheet or **Native text boxes** to use a text box format. The **Undefined** value means the software uses the default output format for the report (Excel spreadsheet).

Report Justification

Specifies the justification of the report window. Select **Top-Left**, **Top-Right**, **Center-Center**, **Bottom-Left**, or **Bottom-Right** to align the report to one of these positions. For example, if you select **Top-Right**, the top-right corner of the report window is aligned to the top-right corner of the view. The **Center-Center** option is the only justification option that scales the report window to fit the view. The other justification options do not scale to the view. For example, if you choose **Top-Right** and the report window is either larger or smaller than the view, it is not resized automatically to fit the view.

NOTE The Report Output Format setting must be set to Native text boxes for justification to function. Otherwise, the justification setting is automatically set to Center-Center.

After creating the view and defining its properties, use the **Associate Objects to View** command to specify the content of the view.

The **SP3DConvertExcelEmbedded.dll** is a delivered custom command that allows you to convert an Excel spreadsheet report to the native text box format for use in 3D Drawings. For information on converting Excel spreadsheet reports, see *Convert Excel Spreadsheet Reports to Native Text Box Format Custom Command* (on page 403).

See Also

Place Report View Command (on page 319)

Select Template Dialog Box

Specifies a template to associate with the selected view.

This dialog box appears when you click **More** in the **Value** dropdown on the **Report Properties** dialog box while placing a new report view on the drawing sheet. For more information, see *Place Report View Command* (on page 319).

Click **OK** to return to the **Report Properties** dialog box.

See Also

Place Report View Command (on page 319)
Report Properties Dialog Box (Place Report View Command) (on page 320)

Reports Compatible with Drawings

There are three conditions that must be satisfied before a report can be successfully embedded in a drawing and produce useful results.

- Pure SQL queries are not supported
- Report must have a column named OID and LocatableOID
- There must be a column named ItemNum

The following reports can be embedded in a drawing successfully. The product allows you to embed any of the delivered reports in a drawing, but the following reports provide more meaningful results when embedded in a drawing. The following reports are the only reports that are compatible with the bubble label (Reference_Circle_CA_L) in drawings. For more information on each report, see the *Smart 3D Reports User's Guide*.

PipeRuns by Drawing (Deliverables\PipeRuns by Drawing)

Equipment Material Take-Off (Equipment)

Solids of Design Equipment with Composite CoG (Equipment)

Hangers and Supports for Components - Drawings (Hangers and Supports)

Hangers and Supports for Supported Route - Drawing (Hangers and Supports)

Hangers and Supports for Supporting Structure - Drawing (Hangers and Supports)

Hangers and Supports End View - Drawing (Hangers and Supports)

Hangers and Supports ISO View - Drawing (Hangers and Supports)

Hangers and Supports Side View - Drawing (Hangers and Supports)

Hangers and Supports Pipe Rack Sorted by Name - Export to Icarus (Hangers and Supports)

Equipment Piping Trim (Piping)

HVAC Material Take-Off (HVAC)

Piping Fittings - Export to Icarus

Piping Material Take-Off (Piping)

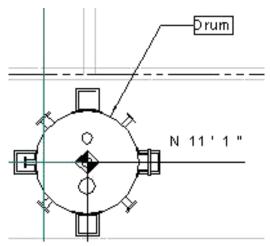
Piping Pipe Line List (Piping)

Place a Label Command

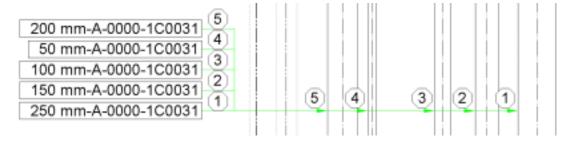
Manually labels items on a drawing. This command is available when you are editing a 3D drawing in SmartSketch Drawing Editor.

You may want to manually place labels to identify or emphasize objects beyond what is provided by automatic labeling. You can choose one of the delivered label rules, or you can customize your own label rule.

You can select a single object and place a label for the object.



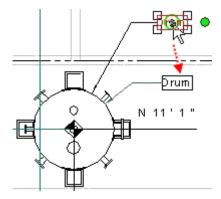
With an applicable label rule, you can also select multiple objects, and then place a group of labels. The following figure shows grouped labels aligned to each other and consolidated by label types with optional leaders and bubble labels. For more information, see *Group Labels* (on page 326).



When you place labels using this command, the software automatically saves the labels to the **DwgTemplate** layer so they are preserved when you update the drawing document.

TIPS

 To move a label after it has been placed, click and drag the label by its origin point, identified by the green circle in the middle of the label object. In the following example, the Drum label is moved.



Snapping behavior is automatic. In the previous example, if you click and drag the **Drum** label by its origin point and hover over another label, then release the **Drum** label, it snaps to the new position.

Place a Label Ribbon (on page 323)

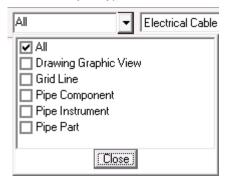
Place a Label Ribbon

Sets options for manual label placement on a drawing. To access this ribbon, click **Place a Label** .

■ NOTE When you place labels using this command, make sure you are on the **DwgTemplate** layer of the drawing so the software preserves the manually placed labels when you update the drawing.

Object Selection Filter

Specifies what type of objects can be selected in the drawing when placing a label. The list displays only object types in the drawing. Select one or more object types, or select **All** to select all object types. Click **Close** to close the list. **All** is the default value.



Label Name

Specifies a label rule. The label rule defines the label type:

- A single or a group label
- Inclusion and type of leaders
- Inclusion and type of bubble labels

Label rules are located in the [Reference Data

Folder]\SharedContent\Drawings\Catalog\Rules\LabelRules folder.

TIP Click More to select a label rule from a hierarchical list.

Font

Defines the text font in the label.

Text Size

Defines the text size in the label.

Text Color

Defines the text color in the label.

B Bold

Applies bold formatting to the label text.

I Italic

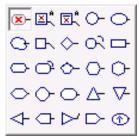
Applies italic formatting to the label text.

U Underline

Underlines the text.

Shape

Defines a shape for the label border.



∰ / ∰Ĵ Orientation

Specifies the positioning and direction of text in the label. Select **Horizontal** $\stackrel{\mathbb{RE}}{\longleftrightarrow}$ or **Vertical**

/ Leaderline On/Off

Specifies whether you want a leader line pointing to the object.

Leaderline Jog On/Off

Specifies whether a jog in the leader line is acceptable.

Leader Color

Defines the color of the leader line. The leader line color can be different than **Text Color**

் / **்** Boundary

Specifies whether the leader terminator originates inside or at the boundary of the object. Select **Boundary** \Box or **Inside** \Box .

Tolerance Zone

Specifies the required distance between the labeled object and the label before a leader appears. For example, if the leader tolerance value is set to 10 mm, then the label must be at least 10 mm away from the labeled object before a leader appears.

🏥 / 🧺 / 🥳 / ry Dimensioned Label

Places a label in a dimension format. Choose from **Distance Between** [**], **Coordinate Dimension**, **SmartDimension** [**], and **Angle Between** [**]. A label displays instead of a dimension value. For more information, see *Dimensioned Label Command* (on page 335).

Pipe BOP

Places a label that displays the elevation or inverted elevation value of a pipe route object. For more information, see *Elevation Label Command* (on page 337).

🗖 As Drawn

Specifies that the label be shown using default values in the label rule. The software ignores font, text size, text color, and other formatting options.

Place a Manual Label

- 1. Right-click a drawing, and click **Edit** on the shortcut menu. The drawing opens for editing in **SmartSketch Drawing Editor**.
- 2. On the **Drawings Compose** toolbar, click **Place a Label**
- Click Object Selection Filter and select the types of objects available for selection. For example, selecting Physical Connections only allows you to select physical connections. Click Close.
- 4. Select an object in a drawing view to label. If you are placing a group label, select two or more objects in a drawing view.
- 5. In the Label Name box, select a label rule.
 - Select a label rule applicable to the selected objects.
 - To pick from a hierarchical list:
 - i. Select More.

The **Select Label** dialog box displays.

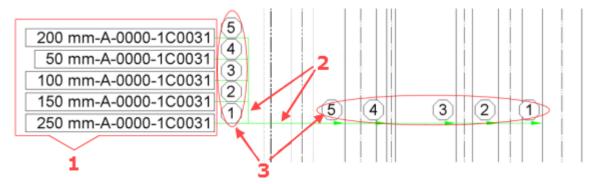
ii. Select a label template folder in the left pane of the dialog box, select a specific label name in the right pane, and then click **OK**.

- With an applicable label rule, you can place a group label. For more information, see Group Labels (on page 326).
- 6. Make the required typeface selections in the Font, Text Size, and Text Color boxes.
- 7. Click **Bold B**, **Italic I**, and **Underline U** to achieve the required formatting.
- 8. Click **Shape S**, and select the shape of the label border.
- 10. Make the required leader line selections in the **Leaderline On/Off** /, **Leaderline Jog On/Off** , and **Leader Color** boxes.
- 11. Click **Boundary** deto clip the leader at the object boundary, or click **Inside** deto extend the leader to the inside of the object.
- 12. Type a value in the **Tolerance Zone** box. This value determines the distance required for a leader to display.
- 13. Select a dimension label in order to place a dimension-like label. For more information, see *Dimensioned Label Command* (on page 335).
- 14. Select an elevation label by if you want to place a label that displays the elevation of a piping object. For more information, see *Elevation Label Command* (on page 337).
- 15. Select **As Drawn** if you want the label appearance to reflect the label rule definition only, ignoring any formatting overrides.
- 16. Click the drawing to place the label.
 - **TIP** During placement, the label will automatically align to the left or right side of an existing label. You can also align the label to any object keypoints in the drawing. For more information on using alignment settings, see *Align Dimensions Command in* the *SmartSketch Drawing Editor Help*.
- 17. Continue placing labels on the drawing if required.
- 18. Right-click to end the command.
- 19. Save your changes before exiting SmartSketch Drawing Editor.
- NOTE You can remove manual edits from a drawing using Clear Manual Edits in SmartSketch Drawing Editor. For more information, see Clear Manual Edits Command (on page 379).

Group Labels

■ NOTE The software delivers Name_None_CA_JL as an example label rule that places single or group labels. See your administrator for other customized group label rules that may be in your catalog.

A group label allows selection of multiple objects and groups the labels for the objects. Optionally, the group label also places leaders and bubble labels.



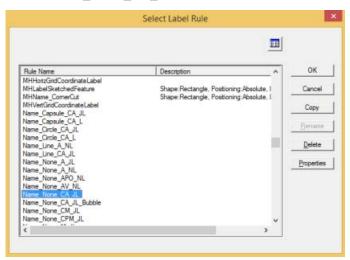
- 1 Group label
- 2 Leaders
- 3 Bubble labels

You can add a group label as an option to any manually placed label. The **Place a Label** command places a single label when you select one object or a group label when you select multiple objects. The software delivers the **Name_None_CA_JL** label template as an example implementation of single and group labels. You can find the Name_None_CA_JL.xml file in the [Reference Data Folder]\SharedContent\Drawings\Catalog\Labels\Templates folder.

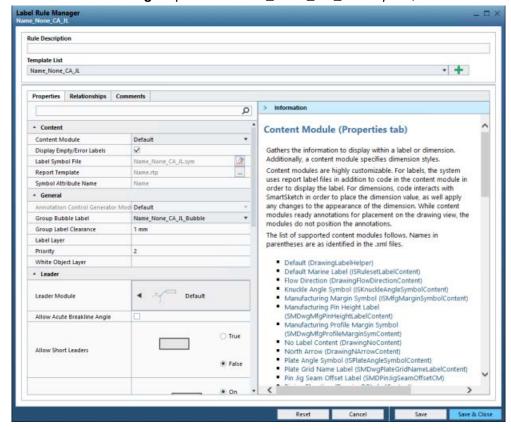
Defining a group label

■ NOTE You must have already defined the view style type and the View Style Properties dialog box must be displayed. For more information, see Define View Style Dialog Box and View Style Properties Dialog Box.

Select Name_None_CA_JL in the Select Label Rule dialog box, as shown below.



2. Click Properties.



The Label Rule Manager opens the Name None CA JL template, as shown below.

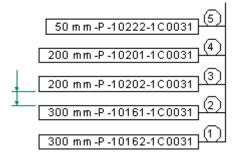
- To change the distance between grouped labels, type a new value in the Group Label Clearance box.
- To suppress the bubble the software draws around grouped labels, select NONE from the Group Bubble Label list.
- 5. Click Save & Close to exit the Label Rule Manager.

Group Bubble Label

Defines the bubble label template and symbol used by the group label. If you select **NONE**, then the software places the group labels without bubble labels.

Group Label Clearance

Specifies the distance between labels in the group.



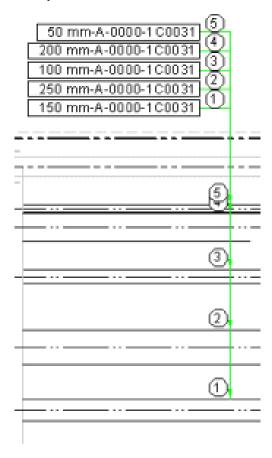
Leaders

Leaders jog as needed based on the position of the label group to the objects, as shown in the following examples.

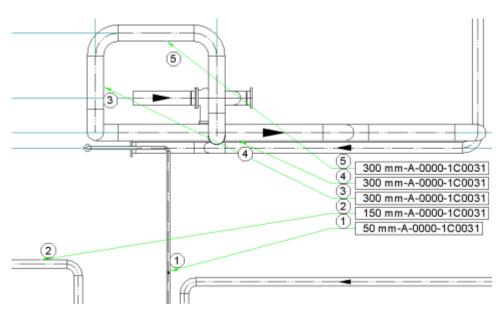
Group label with horizontal leader



Group label with vertical leader



Group label with fanned leader



Group label with no leader

Duct Banks_	Conduit Run-1-41
Duct Banks_	Conduit Run-1-42
Duct Banks_	Conduit Run-1-43
Duct Banks_	Conduit Run-1-44
Duct Banks_	Conduit Run-1-45
Duct Banks_	Conduit Run-1-36
Duct Banks	Conduit Run-1-37
Duct Banks_	Conduit Run-1-38
Duct Banks_	Conduit Run-1-39
Duct Banks_	Conduit Run-1-40
Duct Banks_	Conduit Run-1-31
Duct Banks	Conduit Run-1-32
Duct Banks_	Conduit Run-1-33
Duct Banks_	Conduit Run-1-34
Duct Banks_	Conduit Run-1-35

Sorting and Alignment

The software sorts the labels according to the relative location of the objects (as defined by the object center points) and the type of leaders.

Leader Type	Sorting	
Horizontal	Left to right	
Vertical	Top to bottom	
Fanned	To minimize crossing of leaders	

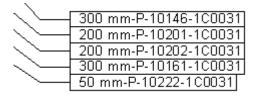
Leader Type	oe Sorting	
None	Top to bottom or left to right	

Labels align to the leader connection side.

Right side alignment

50 mm-P-10222-1C0031	
300 mm-P-10146-1C0031	
200 mm-P-10201-1C0031	
200 mm-P-10202-1C0031	
300 mm-P-10161-1C0031	

Left side alignment



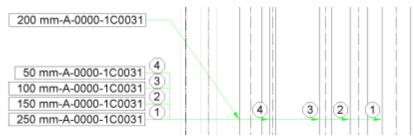
Selecting multiple objects for the group label

Use one of the following methods to select objects in the SmartSketch Drawing Editor.

- Press CTRL and select each object individually in a drawing view.
- Click and drag in a drawing view to create a selection window. The software selects all objects completely within the selection window.
- Press CTRL and then click and drag in a drawing view to create a line selection. The software selects all objects overlapping the line.

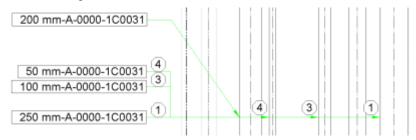
Moving a label in a group

You can select and move a label to separate it from the group label.



Deleting a label in a group

You can select and delete a label. The software also deletes the leader and bubble labels, but the remaining bubble labels are not renumbered.



Limitations

Group labels have the following label parameter limitations:

- Orientation ♣ / ♣ is ignored. Group label text is always horizontal.
- Boundary 🗗 / 🗹 is ignored.
- Tolerance Zone is ignored.

The following values for **Shape** cannot be used as the label border: **Cloud** \bigcirc , **Pentagon** \bigcirc , **Triangle Up** \triangle , and **Triangle Down** ∇ .

For more information on label parameters, see *Place a Label Ribbon* in *Place a Label Command* (on page 322).

Place a Manual Weld Label

- 1. Right-click a drawing, and click **Edit** on the shortcut menu. The drawing opens for editing in **SmartSketch Drawing Editor**.
- 2. On the toolbar, click **Manually Place Labels**.
- (This step is only for marine mode) Use the **Object Filter** to enable only certain objects for selection. For example, selecting **Physical Connections** in the Object filter list only allows you to select physical connections.
- 4. In the Label Name drop-down menu, select a label rule.
 - TIP The label rules are located on the application server in the \Symbols\Drawings\Catalog\Labels\Templates folder.
- 5. To pick from a hierarchical list, select More.

The Select Label dialog box displays.

- 6. Select a label template folder in the left pane of the dialog box, select a specific label name in the right pane, and then click **OK**.
- 7. Click on a physical connection.

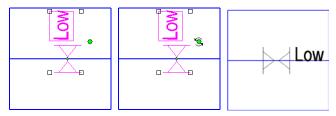
The correct weld symbol automatically displays based on the physical connection properties.

8. In the **Font** box, specify the font.

- 9. In the **Text size** box, specify the size of the text.
- 10. Click **Text Color** and select a color for the label text.
- 11. Click the **Bold B**, **Italic I**, and **Underline** Ubuttons to achieve the necessary formatting.
- 13. Select a leader and leader jog.
- 14. Click **Leader Color** and select a color for the leader.
- 15. Choose the **Boundary** option to clip the leader at the object boundary, or choose the **Inside** option to extend the leader to the inside of the object.
- 16. Key in a value for the **Leader Tolerance** value. This value determines the distance required for a leader to display. For example, if the leader tolerance value is set to 10mm, the label must be at least 10mm away from the labeled object before a leader displays.
- 17. Choose a think dimension label in order to place a dimension that acts as a label. For more information, see *Dimensioned Label Command* (on page 335).
- 18. Select **As Drawn** if you want the label appearance to reflect the label definition only and not have any formatting overrides.

TIPS

- If you select **As Drawn**, the label appearance is determined by the label definition. You cannot set the font, text size, text color, and so forth for the label.
- Click the drawing to place the label. If you hover directly over the physical connection, the label will snap to the physical connection line, but will not align.
- To move the label after it has been placed, click and drag the label by its origin point, identified by the green circle in the middle of the label object. In the graphic below, the weld symbol is moved.
- To rotate the label after it has been placed, select the label and click the green circle located on the outside of the label. Drag the green circle until the label is aligned correctly to the physical connection.



- 19. Continue placing labels on the drawing if necessary.
- 20. Right-click to end the command.
- 21. Save your changes before exiting **SmartSketch Drawing Editor**.

■ NOTE You can remove manual edits from a drawing using **Clear Manual Edits** in **SmartSketch Drawing Editor**. For more information, see *Clear Manual Edits Command* (on page 379).

See Also

Place a Manual Label (on page 325)

Place a Manual Flow Arrow on an Orthographic Drawing

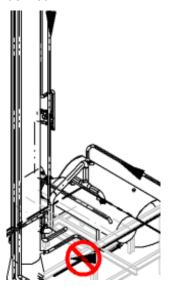
You can place flow arrows on pipe objects in orthographic drawings with the **Place a Label** command in **SmartSketch Drawing Editor**.

You can also add flows arrows as part of the drawing view style definition. For more information, see *Add Flow Arrows to Orthographic Drawings* in the *Drawings and Reports Reference Data Guide*.

- 1. Right-click a drawing, and click **Edit** on the shortcut menu. The drawing opens for editing in **SmartSketch Drawing Editor**.
- 2. On the toolbar, click Manually Place Labels **.
- 3. Click an object in a drawing view to label.
- 4. In the Label Name drop-down menu, select More to display the Select Label dialog box.
- 5. Select a flow label template folder in the left pane of the dialog box, select a specific label name in the right pane, and then click **OK**.
- 6. Select a pipe object in the drawing to place the label.



If the arrow has a red-crossed circle over it, the selected pipe does not have flow direction defined.



7. Right-click to end the command.

8. Save your changes before exiting SmartSketch Drawing Editor.

■ NOTES

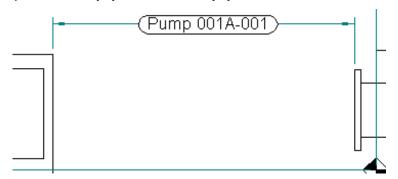
- To move the flow arrow after it has been placed, click and drag the arrow by its origin point, identified by the green circle in the middle of the flow arrow object.
- You can remove manual edits from a drawing using Clear Manual Edits

 SmartSketch Drawing Editor. For more information, see Clear Manual Edits Command (on page 379).
- Centerlines must display for pipelines before flow arrows can be placed.

Dimensioned Label Command

Places a label with dimension formatting. The dimension displays the label instead of a dimension value. **Dimensioned Label** is an option on the **Place a Label** ribbon. You can select **Distance Between** Coordinate Dimension SmartDimension on the formatting of each dimension type, see *Distance Between Command*, Coordinate Dimension Command, SmartDimension Command, and Angle Between Command in the SmartSketch Drawing Editor Help.

Before selecting **Dimensioned Label**, you must select an appropriate label from **Label Specification** on the **Place a Label** ribbon. The following example shows a dimensioned label on a composed drawing. A **Distance Between** indicated dimension is placed between two pumps with the **Equipment Plane Equipment Name** label selected.



■ NOTE Only two objects can be selected with **Dimensioned Label**. The label placed is determined by the first object you select, not the second.

See Also

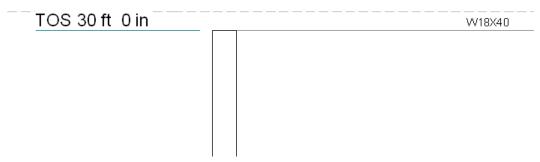
Place a Dimensioned Label (on page 336) (Marine mode only) Place a Dimensioned Label (on page 335) (Plant mode only)

Place a Dimensioned Label

This workflow places a dimensioned label on a composed drawing.

- Right-click a drawing, and click Edit on the shortcut menu.
 The drawing opens in SmartSketch Drawing Editor.
- 2. On the toolbar, click Manually Place Labels.

- 3. In the Label Name drop-down menu, select the Structural Framing Elevation_Grid Line TOS label from the delivered folder.
 - **NOTE** To pick from a hierarchical list, select **More**.
- 4. Select Coordinate Dimension from the dimension label drop-down menu. The dimensioned label ribbon displays.
 - **NOTE** You can select any of the dimension commands from the drop-down menu.
- 5. Select a structure member to label.
- 6. Click to place the Coordinate Dimension label.



- TIP You can use custom dimension styles when placing dimension-like labels. For more information, see *Dimension Properties Dialog Box* in the *SmartSketch Drawing Editor User's Guide*.
- 7. Continue placing labels if necessary.
- 8. Right-click to end the command.
- 9. Save your changes and exit SmartSketch Drawing Editor.

See Also

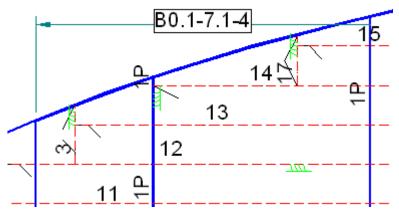
Dimensioned Label Command (on page 335) Place a Manual Label (on page 325)

Place a Dimensioned Label

This workflow demonstrates the ability to place a dimensioned label on a marine mode Steel Order drawing by placing a dimension-like label on two edges of a plate. For more information on dimensioned labels, see *Dimensioned Label Command* (on page 335).

- 1. Right-click a Steel Order drawing, and click **Edit** on the shortcut menu. The drawing opens in SmartSketch Drawing Editor.
- 2. On the toolbar, click Place a Label ...
- 3. Use the **Object Filter** to enable only **Plate Parts** for selection.
- 4. In the **Label Name** drop-down menu, select the **Block Parent** label from the delivered Steel Order folder.
 - NOTE To pick from a hierarchical list, select More.
- 5. Select the **Distance Between** dimension label command from the drop-down menu. The dimensioned label ribbon displays.

- 6. Select the edge of a plate that you want to label.
 - NOTE The first selected object is labeled, not the second.
- 7. Select a second edge. For example, select the opposite edge of the plate you selected.
 - NOTE The second point only determines how long the dimension line will be.
- 8. After selecting the second object, the correct label will display.



TIP You can use custom dimension styles when placing dimension-like labels. For more information, see *Dimension Paper Space Objects for 3D Drawings* (on page 268) in the *SmartSketch Drawing Editor User's Guide*.

- 9. Continue placing labels on the drawing if necessary.
- 10. Right-click to end the command.
- 11. Save your changes before exiting SmartSketch Drawing Editor.

See Also

Dimensioned Label Command (on page 335)
Place a Manual Label (on page 325)

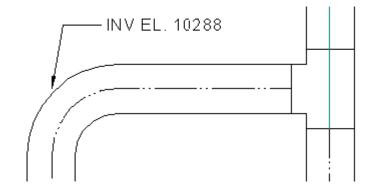
Elevation Label Command

Pipe BOP places a label that displays the elevation or inverted elevation value of the route object. For more information, see *Elevation versus Invert Elevation* below.

The **Pipe BOP** is available only if you select one of the following templates:

- Piping Plan_Routable_Center Elev_M Select to place a label for the center elevation of the pipe.
- Piping Plan_Routable_INV Elev_M Select to place a label for the inverted elevation of the pipe.
- Piping Plan_Routable_WPBOP Elev_M Select to place a label for the working point of the pipe.

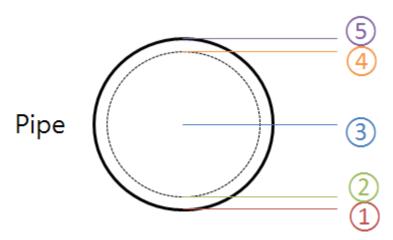
The following example shows an elevation label being used on a composed drawing. An inverted elevation is placed on a pipeline with the Piping Plan_Routable_INV Elev_M template selected.



NOTE Elevation labels support only manual placement and label templates for piping disciplines. To apply these templates to other disciplines, such as HVAC, cable tray, and conduit, modify and use these templates or create a new label template based on these templates.

Elevation versus Inverted Elevation

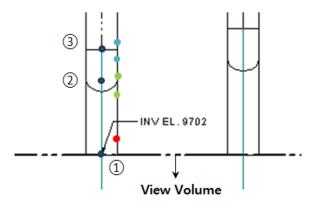
Inverted elevation is the inside elevation of a pipe, HVAC, cable, or conduit route object. Elevation and inverted elevation can be calculated at following locations:



Location	ocation INV (Inverted Elevation)		Elev (Elevation)	
Тор	4	Z of route object + (OutDiameter/2 - Thickness)	5	Z of route object + (OutDiameter/2)
Center	3	Z of route object		Z of route object
Bottom	2	Z of route object - (OutDiameter/2 - Thickness)		Z of route object - (OutDiameter/2)

Leader Point Connection

When you click on the route object the leader point is automatically placed near port point, working point or clipping point, whichever is nearest to where you clicked. For example, if you click at the red point the leader point is moved to nearest clipping point (1). If you click at the green points, the leader point is moved to the nearest working point (2). If you click at the cyan points, then the leader point is moved to nearest port point (3).



Place an Elevation Label

This workflow places an elevation label on a composed drawing.

- 1. Right-click a drawing, and click **Edit** on the shortcut menu. The drawing opens in **SmartSketch Drawing Editor**.
- 2. On the horizontal toolbar, click Manually Place Labels.
- 3. In the **Label Name** drop-down menu, select the needed label. For example, to place an inverted elevation label select **Piping Plan_Routable_INV_Elev_M**.

The Pipe BOP command gets activated after you select the label template.

- NOTE To pick a label template from a hierarchical list, select **More**.
- 4. Select a route object to label.
- 5. Click on any point in the empty space to place the elevation label.
- 6. Continue placing labels if necessary.
- 7. Right-click to end the command.
- 8. Save your changes and exit SmartSketch Drawing Editor.

See Also

Elevation Label Command (on page 337) Place a Manual Label (on page 325)

Group Selected Labels

Groups existing labels so that you can move them as a unit. You can choose to stack the labels directly above one another, or stack them diagonally. This simplifies the final formatting of labels that Smart 3D generates automatically.

Group Selected Labels Ribbon

Select by Labels

Indicates that the software selects labels to form the group. You can select the labels individually, by fence, or by dragging a line through the labels.

Select by Objects

Indicates that the software selects objects to form the label group. You can select the objects individually, by fence, or by dragging a line through the objects.

Labels Selection Filter

Specifies a filter to limit the labels you can select. The default option is **All**, which lets you select any label to add to your group. To refine the filter, select as many of the filters from the list as you need. You can select only those labels that match the filter.

// Leaderline On/Off

Specifies whether you want a leader line pointing to the object.

Leaderline Jog On/Off

Specifies whether a jog in the leader line is acceptable.

Leader Color

Defines the color of the leader line.

Selecting the Labels to Group

You have several options to select the labels to group. If you click **Select by Labels**, the software only lets you select labels. Click **Select by Objects** to select objects.

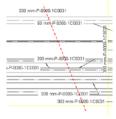
Press CTRL, and then individually click the labels or objects to group.



 Drag a fence around the labels or objects to group. The fence must start and stop in empty space.



 Press CTRL, and then drag a line through the labels or objects to group. The line must start and stop in empty space.



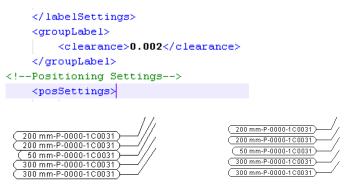
The software places the group of labels without crossing the leader lines, even if the heights of the labels differ, and aligns the labels to the leader connection side.



Left Right

The software sorts the labels so that no leaders in the group cross.

■ NOTE To change the distance between each label in the stack, your administrator can add a <clearance> tag to the label template.



Clearance value of 0.0

Clearance value of 0.002

Group existing labels

- 1. In the Drawings console, right-click a drawing, and click **Edit** on the shortcut menu.
 - The drawing opens for editing in SmartSketch Drawing Editor.
- 2. On the **Drawings Compose** toolbar, click **Group Selected Labels**

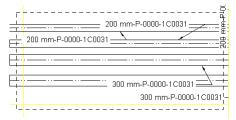
The **Label Filter** displays on the ribbon.

- 3. Click **Select by Labels** or **Select by Objects** to indicate how you want to make the selections for the group.
- 4. If necessary, select filters to refine the **Label Filter**, and then click **Close**.
- 5. Use the tools on the ribbon to specify the leader for the labels. For more information, see *Group Selected Labels* (on page 339).
- 6. Select the labels or objects to group. You can use any of the following techniques:

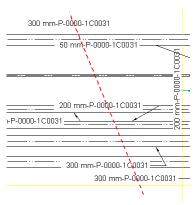
Press CTRL, and then individually click the labels or objects to add to the group.



 Drag a fence around the labels or objects to group. The fence must start and stop in empty space.



 Press CTRL, and then drag a line through the labels or objects to group. The line must start and stop in empty space.

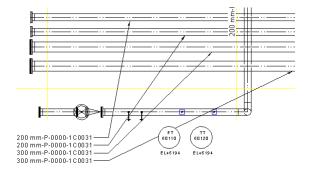


The software displays a blue preview outline to give you an approximation of the size of the stacked labels. The preview moves with the cursor.



7. If necessary, press CTRL to change the preview from stacked to diagonal.

8. Click the location for the relocated labels.

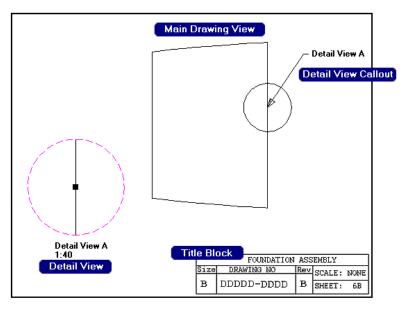


TIP To diagonally space the labels, press CTRL and click to indicate the location.



Place Detail Envelope Command

Creates a detail view for an existing drawing view. Detail views are more than enlargements of the main drawing view. They often contain additional graphical information that is not visible in the main drawing view, such as weld or chalk information. You specify the detail view by drawing a circle or polygon around a portion of the main drawing view. The shape created is called the *detail envelope*. The **Place Detail Envelope** command is only available in **SmartSketch Drawing Editor** when you create a new Composed drawing or open an existing Composed drawing from a 3D task. For example, the following graphic shows the main drawing view and a detail view:

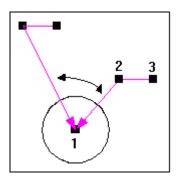


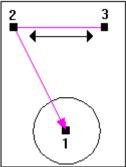
When you select an existing detail envelope, it highlights within the drawing view and the **Edit** ribbon appears so you can make modifications to the selected cutting plane. To place a detail envelope, select a drawing view and click the **Place Detail Envelope** (a) command.

Callouts

Detail envelopes use callouts, or labels, that match the caption for the associated detail view. By default, the command places callouts in a fixed position on the main drawing view when you place the detail envelope. You can click-and-drag the callout to a new position, if necessary.

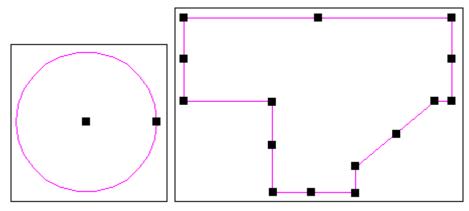
Callouts do not highlight when you select the detail envelope. You select callouts separate from the detail envelope.





Handles

Handles help you modify the shape and placement of the detail envelopes. Circle detail envelopes have two handles, a center point and an edge handle. The center point handle allows you to move the circle, while the edge handle allows you to change the size and shape of the circle. Polygon shapes have handles at each vertex. You can use the handles to move and change the shape of the polygon shape.



After you modify a detail envelope, click **Finish** to save the changes.

Delete Behavior

To delete a detail envelope, select it and press **Delete**. Deleting the detail envelope does not affect the drawing view. However, if there is an associated detail view, a message displays, asking if you want to convert the detail view to a normal drawing view or delete it with the detail envelope.

NOTE You can place detail envelopes inside section views and detail views.

See Also

Place Cutting Plane Ribbon (on page 347)
Place Detail Envelope Command (on page 343)
Place Detail Envelope Ribbon (on page 345)
Edit Detail Envelope Ribbon (on page 345)

Place Detail Envelope Ribbon

Sets options when placing a new detail envelope. This ribbon displays when you select a drawing view, then click **Place Detail Envelope** (9) in the toolbar area.

If you are editing an existing detail envelope, the **Edit Detail Envelope** ribbon appears. For more information, see *Edit Detail Envelope Ribbon* (on page 345).

Circle

Specifies that you want to draw a round detail envelope shape.

Polygon

Specifies that you want to draw a polygon detail envelope shape.

Reference mark

Displays the reference text below the detail view and on the detail view callout on the main drawing view.

Additional callout text

Provides a second line of text for the detail view callout.

See Also

Place Detail Envelope Command (on page 343)

Edit Detail Envelope Ribbon

Sets options when editing an existing detail envelope. This ribbon displays when you select a detail envelope on the drawing sheet, then click **Place Detail Envelope** in the toolbar area. If you have more than one detail envelope selected, this ribbon is disabled.

If you are placing a new detail envelope, the **Place Detail Envelope** ribbon appears. For more information, see *Place Detail Envelope Ribbon* (on page 345).

Reference mark

Displays the reference text below the detail view and on the detail view callout on the main drawing view.

Additional callout text

Provides a second line of text for the detail view callout.

■ NOTE The Reference mark and Additional callout text always show the current values for the detail envelope. The fields are disabled if the detail envelope is associated with a detail view.

See Also

Place Detail Envelope Command (on page 343)

Place a Detail Envelope

The following steps describe the basic workflow for placing a detail envelope on a drawing view.

- 1. In the 3D application, edit a drawing document. The drawing opens in **SmartSketch Drawing Editor**.
- 2. In the drawing document, select a drawing view. Zoom in on the area of the main drawing view to the location you want to place the detail envelope geometry.
- 3. In the toolbar area, click **Place Detail Envelope** . The **Place Detail Envelope** ribbon appears in the toolbar area.
- 4. Click **Circle** O or **Polygon** So you can begin placing points to define the detail envelope. For more information, see *Place Detail Envelope Ribbon* (on page 345).
- 5. Click points as needed to create the detail envelope geometry. If you are creating a polygon shape for your detail envelope, complete the polygon by moving the cursor over the starting point until the **Close Polygon** symbol appears, then click to close the polygon.

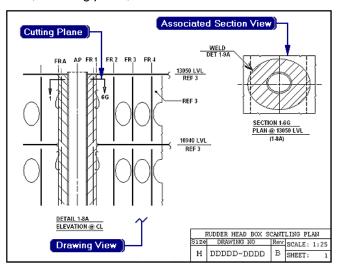
Edit a Detail Envelope

To edit an existing detail envelope, select the detail envelope. The **Edit Detail Envelope** ribbon appears so you can edit the settings on the detail envelope.

For more information on working with detail envelopes, see *Place Detail Envelope Command* (on page 343).

Place Cutting Plane/Section View Command

Creates a cutting plane on a drawing view. The cutting plane is an annotation marker that indicates where to slice a group of objects and from which direction to look at that slice. Cutting planes are made of one or more line segments. When comprised of multiple line segments, cutting planes are also referred to as "jogged" cutting planes. The **Place Cutting Plane** command is only available in **SmartSketch Drawing Editor** when you create a new drawing or edit an existing drawing from a 3D task. For example, the following graphic shows a drawing view, a cutting plane, and a derived section view:



You can only create section views from cutting planes drawn on valid graphical views. Valid graphical views include normal drawing views, section views, detail views, and snapshot views, but exclude keyplan views. Valid graphical views must already be associated to a volume in the model and updated at least once). If you delete a drawing view that has cutting planes, the cutting plans are also deleted. A confirmation message asks if you want to convert associated section views to normal drawing views or delete them as well.

After a cutting plane has been defined, a drawing view appears on your cursor. Click to place the view on the drawing sheet. For more information, see *Place a Cutting Plane/Section View* (on page 350).

When you select a cutting plane, it highlights within the drawing view and the **Edit** ribbon appears so you can make modifications to the selected cutting plane. To place a cutting plane, select a valid drawing view and click the **Place Cutting Plane** command. You can also click the command first and then select the view.

Single-segment and Jogged cutting planes have handles at the end-points, mid-points, and depth-points. You can change the geometry of the selected cutting plane by dragging its handles. Dragging a mid-point handle moves the associated line segment, maintaining its length and slope while altering any adjacent line segments. Dragging a vertex or end-point alters the length and/or slope of the attached line segment(s). Dragging a depth handle alters the depth of the associated section view.

You can move a single-segment cutting plane by clicking and dragging its mid-point handle to the new position. You can move a jogged cutting plane by clicking on a point of the line segment and dragging it; do not click a handle to move the jogged cutting plane.

After modifying a cutting plane, click **Finish** to save the changes. Selecting the **Update section** check box will cause an update of the related section view when the **Finish** button is clicked. The **Update section** option is remembered globally for all section views.

To delete a cutting plane, select it and press the DELETE key. Deleting the cutting plane does not affect the drawing view, but if there is an associated section view, you are asked if you want to convert the section view to a normal drawing view or delete it as well.

Shortcut Menu

When you select an existing cutting plane, you can right-click to get the cutting plane shortcut menu. The shortcut menu allows you to make the following modifications to the cutting plane: **Bring to Front**, **Send to Back**, **Pull Up**, and **Push Down**.

See Also

Place Cutting Plane Ribbon (on page 347)
Place a Cutting Plane/Section View (on page 350)
Section View Orientation Rules in Drawings by Rule (on page 409)

Place Cutting Plane Ribbon

Sets options when placing a cutting plane. This ribbon displays when you select a drawing view. Click **Place Cutting Plane** riangle command to place a new cutting plane.

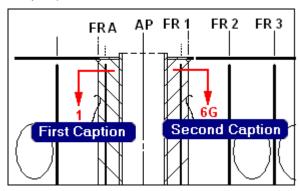
Reference 1

Displays the text shown for the first arrow of the cutting plane.

Reference 2

Displays the text shown for the second arrow of the cutting plane.

■ NOTE The Reference 1 is always assigned to the point of the cutting plane that you placed first, while the Reference 2 text is always assigned to the point of the cutting plane that you placed last.



Depth

Sets the depth of the cutting plane. This box only accepts positive values.

See Also

Place Cutting Plane/Section View Command (on page 346) Place Section View Ribbon (on page 348) Place a Cutting Plane/Section View (on page 350)

Place Section View Ribbon

Sets options when placing a section view. This ribbon displays after you have completed the placement of a cutting plane.

Update section

When checked, the section view updates after it is placed. This option is unchecked by default, but the last setting is saved as a session file preference.

View Style

Specifies a view style, which includes rules for filters, updates, and graphics. The view style controls the output characteristics of the view on the generated drawing. The list displays the 10 most recently used view styles in the session. Click **More...** to display the **Select View Style** dialog box.

■ NOTE Changing the view style for the section view can result in symbology different from that used in the parent view.

View Scale

Sets the scale family and scale used for the section view. The default scale setting is inherited from the parent view.

See Also

Place Cutting Plane/Section View Command (on page 346) Place Cutting Plane Ribbon (on page 347) Place a Cutting Plane/Section View (on page 350)

Edit Cutting Plane Ribbon

Sets options when editing a cutting plane. This ribbon displays when you select an existing cutting plane that is associated with a section view.

Finish

Saves the changes to the cutting plane. If the cutting plane is associated with a section view, the section view contents change if the **Update section** check box is selected.

Update section

When checked, the section view updates after it is placed. This option is not checked by default, but the last setting is remembered as a session file preference.

Reference 1

Displays the text shown for the first arrow of the cutting plane.

Reference 2

Displays the text shown for the second arrow of the cutting plane.

Depth

Sets the depth of the cutting plane. This box only accepts positive values.

See Also

Place Cutting Plane/Section View Command (on page 346)
Place Cutting Plane Ribbon (on page 347)

Edit Section View Ribbon

Sets options when editing a section view. This ribbon displays when you select a section view.

Properties

Opens the **Drawing View Properties** dialog box, allowing you to change the properties of a drawing view. For more information, see *Drawing View Properties Dialog Box (Place View Command) - Steel Order Drawings* (on page 281).

Finish

Saves the changes to the section view.

Update section

When checked, the section view updates when you select **Finish**. This option is unchecked by default, but the last setting is remembered as a session file preference.

View Style

Specifies a view style, which includes rules for filters, updates, and graphics. The view style controls the output characteristics of the view on the generated drawing. The list displays the 10 most recently used view styles in the session. Click **More...** to display the **Select View Style** dialog box.

■ NOTE Changing the view style for the section view can result in symbology different from that used in the parent view.

View Scale

Sets the scale family and scale used for the section view. The default scale setting is inherited from the parent view.

Rotate

Rotates the section view clockwise by multiples of 90 degrees. When the view is rotated, user-placed and user-modified dimensions in the section view are deleted.

NOTE To rotate a section view, you must bulkload the following files to the database:

- [Product Folder]\CatalogData\BulkLoad\AdditionalDataFiles\Delta2009.1to2011\1_AMD_Delta_2 009.1_2011_Drawings.xls
- [Product Folder]\ShipCatalogData\BulkLoad\AdditionalDataFiles\DeltaSMV2011toV2011SP1\1_A MD_Delta_V2011_2011SP1_SM_ShipDrawings.xls. (Marine mode only)

See Also

Place Cutting Plane/Section View Command (on page 346) Place Section View Ribbon (on page 348)

Place a Cutting Plane/Section View

Follow the general steps below to place a cutting plane on a drawing view.

- In a 3D application, edit a drawing document. The drawing opens in SmartSketch Drawing Editor.
- In the drawing document, select a drawing view.
- 3. In the toolbar area, click **Place Cutting Plane** . The **Place Cutting Plane** ribbon appears in the toolbar area. For more information, see *Place Cutting Plane Ribbon* (on page 347).
- 4. On the **Place Cutting Plane** ribbon, type the names for the reference text string you want shown on the cutting plane.
- 5. Click the first point in the line string, then proceed selecting as many points as you need to define the cutting plane geometry.
 - NOTE Jogged cutting planes are not supported in Ruleset drawings.
- 6. Right-click when you have finished defining the cutting plane line segments and are ready to define the depth of the cutting plane.
- 7. To change the view direction, move the cursor to the appropriate side of the cutting plane. The direction automatically changes according to which side of the cutting plane segment the cursor is on.
- 8. Click to define the section depth and view direction. You can also key in a value in the **Depth** field and click to define the view direction. The value displayed in the **Depth** field is defined in model space, not paper space.

The drawing view automatically appears on your cursor.

■ NOTES

 Selecting the **Update section** check box will cause an update of the related section view when the view is placed. This option is not checked by default, but the last setting is saved as a session file preference.

- The section view inherits the scale from the parent view by default.
 - TIP You can change the View Style or View Scale before placing the view.
- 1. Click to place the view on the sheet.

Edit a Cutting Plane

To edit an existing cutting plane, select the cutting plane. The Edit Cutting Plane ribbon appears so you can edit the settings on the cutting plane. For more information, see Edit Cutting Plane Ribbon (on page 349).

Edit a Section View

 To edit an existing section view, select the section view. The Edit Section View ribbon appears so you can edit the settings on the section view. For more information, see Edit Section View Ribbon (on page 349).

Delete a Cutting Plane or Section View

- To delete a cutting plane, select it and press the DELETE key. If the cutting plane has an associated section view, a confirmation box appears allowing you to either convert the section view to a normal drawing view or delete the section view with the cutting plane.
- To delete a section view, select it and press the DELETE key. The associated cutting plane line remains, but only as a paper space graphic.

For more information on working with cutting planes, see *Place Cutting Plane/Section View Command* (on page 346).

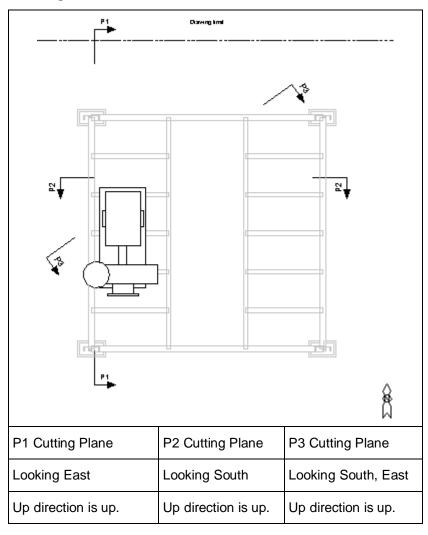
Section View Orientation Rules in Orthographic Drawings

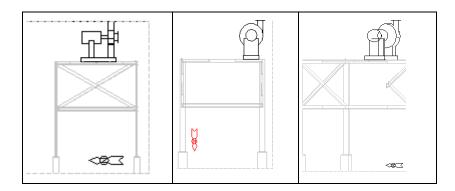
Section View Orientation Behavior

The software automatically determines the "up" direction for section views. In most cases, the up direction is "up," or +Z. In cases where using +Z is impossible (for example, if the section view direction is Looking Plan or Looking Up.), the software orients the view to the North direction.

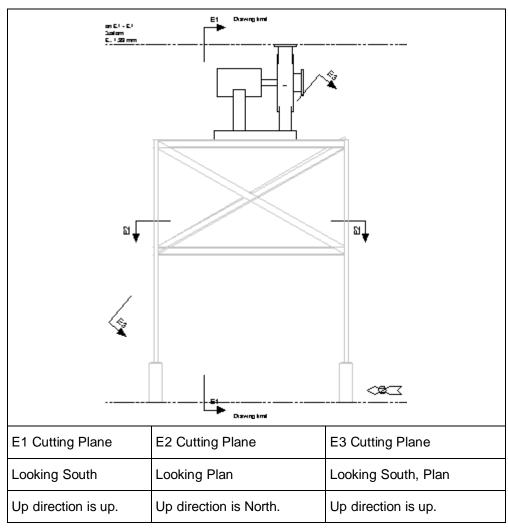
The following examples outline common section view orientations:

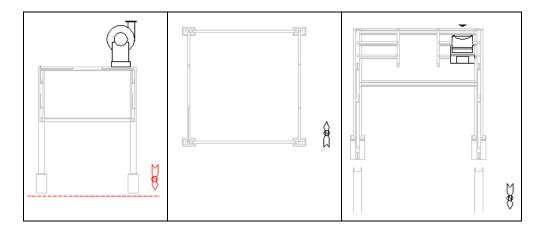
Looking Plan



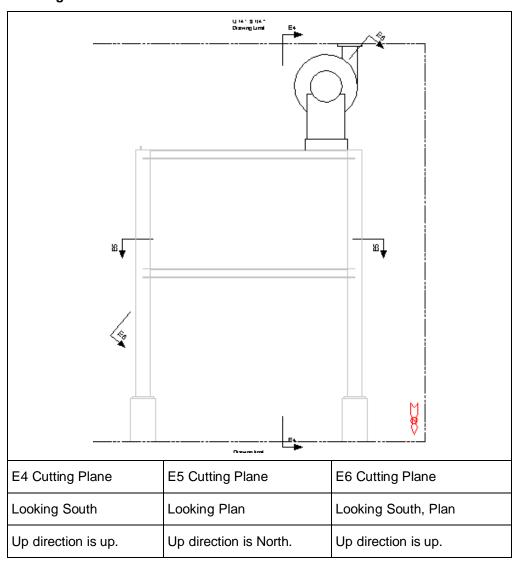


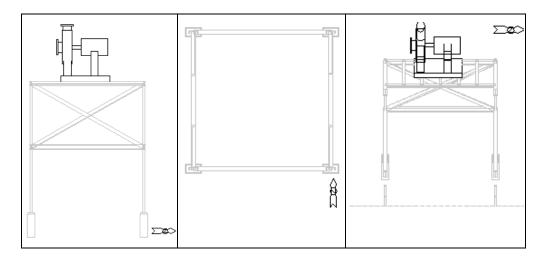
Looking North





Looking East





Place a Section View

The following procedure describes editing a drawing document to add a section view.

Section views are extracted from main drawing views or other section views. Section views are similar to detail views, except that they display information in an orientation that is different from that of the originating view. The cutting plane is an annotation marker that indicates where to slice a needed group of objects and from which direction to look at that slice. The cutting plane is comprised of one or more line segments. When comprised of multiple line segments, cutting planes are referred to as "jogged" cutting planes.

■ NOTES

- For more information on the **SmartSketch Drawing Editor** commands used in this procedure, see *Place Detail Envelope Command* (on page 343) and *Place Cutting Plane/Section View Command* (on page 346) in the *SmartSketch Drawing Editor Help*.
- For information on the toolbar used in this procedure, see *Drawings Compose Toolbar* (on page 275) in the *SmartSketch Drawing Editor Help*.
- For information on the 2D commands available for editing, see the *SmartSketch Drawing Editor Help*.

Place a Cutting Plane

The following steps describe the basic workflow for placing a cutting plane on a drawing view used in creating a section view.

- 1. Open a drawing in SmartSketch Drawing Editor.
- Click Cutting Plane on the toolbar.
- 3. In the drawing document, select a drawing view.
- 4. Type in text for the first arrow of the cutting plane in **Reference 1**. Type in text for the second arrow of the cutting plane in **Reference 2**.
- 5. To define the length of the cutting plane segment, click in the view to define the location of the first point, then click to define the location of the second point.

The cutting plane segment *displays* on the drawing, with arrows showing the default view direction.

- 6. To jog the cutting plane, click to define additional line segments. When you are ready to define the depth, right-click to proceed to the next step.
 - **NOTE** The cutting plane jogging behavior is not available in Ruleset drawings.
- 7. To change the view direction, move the cursor to the appropriate side of the cutting plane.

The direction automatically changes according to which side of the cutting plane segment the cursor is on.

- NOTE The view direction is always perpendicular to the first cutting plane segment.
- 8. To define the section depth, click to define the distance and view direction. You can also key in a value in **Depth** and click to define the view direction.
 - **NOTE** The section depth is defined in model space, not paper space.

The drawing view automatically displays on your cursor.



- NOTE Selecting the **Update section** check box causes an update of the related section view when the view is placed. The **Update section** option is remembered globally for all section views.
- TIP You can change the View Style or View Scale before placing the view.
- 9. Click to place the view on the sheet.
 - **NOTE** The child view inherits the scale from the parent view by default.

Modify the Width, Depth, and Location of a Cutting Plane

The following steps describe the basic workflow for modifying the width of a cutting plane.

- 1. Open a drawing in SmartSketch Drawing Editor.
- 2. Click on an existing cutting plane.

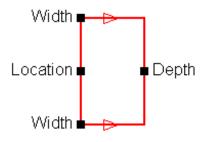
The **Edit Cutting Plane** ribbon displays in the toolbar area.

3. Click and drag the outside handles of the cutting plane segment in order to change its length.

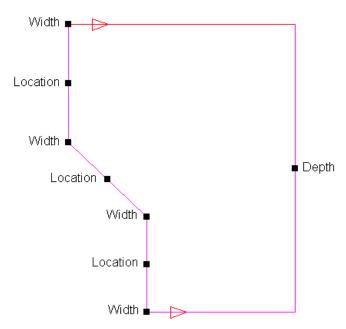
■ NOTES

- The cutting plane handles correspond to the sides of the section view that are opposite to the section view depth handles. Changing the overall length of a cutting plane changes the corresponding sides of the related section view.
- In the case of a jogged cutting plane, each cutting plane segment will have its own modification handles.
- 4. To change the location of the cutting plane segment, click and drag the middle handle that lies between the width handles.
 - **NOTE** To change the location of an entire jogged cutting plane, click and drag the line segment; do not click a handle to move the jogged cutting plane.
- 5. Click and drag the handle opposite of the location handle to modify the cutting plane depth. You can also key in a value in **Section depth** and click in the view to define the view direction.

Single Cutting Plane



Jogged Cutting Plane



■ NOTE Selecting the **Update section** check box causes an update of the related section view when the **Finish** button is selected. The **Update section** option is remembered globally for all section views.

- 6. When the appropriate changes have been made to the cutting plane, select **Finish** in the **Edit Cutting Plane** ribbon.
- 7. In plant mode, right-click the view and select **Update**.

-OR

In marine mode, right-click on the section view in the **Drawings View Explorer** and select **Update**.

Modify the Size of a Section View

The following steps describe the workflow for modifying the size of an existing section view.

1. Open a drawing in SmartSketch Drawing Editor.

2. Click on an existing section view.

The Edit Section View ribbon displays in the toolbar area.



■ NOTE The ribbon allows you to change the View Style and View Scale, as well as edit the View Properties.

- 3. Click and drag the handles on the top and bottom (or left and right sides) of the section view in order to change the view size.
 - NOTE Selecting the **Update section** check box causes an update of the section view when the **Finish** button is selected. The **Update section** option is remembered globally for all section views.
- 4. When the appropriate changes have been made to the volume size, select **Finish** in the **Edit Section View** ribbon.
- 5. In plant mode, right-click the view and select **Update**.

-OR-

In marine mode, right-click on the section view in the **Drawings View Explorer** and select **Update**.

Place Detail View Command

Creates a detail view based on the selected detail envelope. A detail view is extracted from a main drawing view, a section view, or another detail view.

The drawing view style defines how objects appear in the detail views.

■ NOTE Detail views are based on detail envelopes. For more information, see *Place Detail Envelope Command* (on page 343).

Edit Detail View Properties

You can edit the properties associated with the detail view. When you access the properties through one of the following methods, the **Drawing View Properties** dialog box appears. For more information, see *Drawing View Properties Dialog Box (Drawing View Shortcut Menu)* (on page 382).

- Right-click the view and select **Properties** on the shortcut menu.
- Select the view, then select Edit > Properties.

Delete Behavior

To delete a detail view, select it and press **Delete**. Deleting the detail view does not delete the corresponding detail envelope.

See Also

Place Cutting Plane Ribbon (on page 347) Place a Detail View (on page 359) Place Detail View Ribbon (on page 359)

Place Detail View Ribbon

Sets options when placing or editing a detail view. This ribbon displays when you select a detail envelope, then click **Place Detail View**.

For information on placing detail envelopes, see *Place Detail Envelope Command* (on page 343).

Sheet Name

Indicates the drawing sheet where the detail view is placed. By default, the detail view is placed on the same sheet as the main drawing view.

View Style

Specifies a view style, which includes rules for filters, updates, and graphics. The view style controls the output characteristics of the view on the generated drawing. The list displays the 10 most recently used view styles in the session. Click **More...** to display the **Select View Style** dialog box.

■ NOTE Changing the view style for the detail view can result in symbology different from that used in the parent view.

View Scale

Sets the scale family and scale used for the detail view. The default scale setting is inherited from the parent view.

See Also

Place Detail View Command (on page 358)

Place a Detail View

The following procedure steps you through editing a drawing document to add a detail view.

A detail view is extracted from a main drawing view, a section view, or another detail view. Detail views are more than enlargements of the main drawing view. They often contain additional graphical information that is not visible in the main drawing view, such as weld or chalk information. You specify the detail view by drawing a circle or polygon around a portion of the main drawing view. The shape created is called the *detail envelope*.

■ NOTES

- For more information on the SmartSketch Drawing Editor command used in this procedure, see *Place Detail Envelope Command* (on page 343).
- For information on the toolbar used in this procedure, see *Drawings Compose Toolbar* (on page 275).
- For information on the 2D commands available for editing, see the *SmartSketch Drawing Editor Help*.

Place a Detail Envelope

The following steps describe the basic workflow for placing a detail envelope on a drawing to be used in creating a detail view.

NOTE Detail Envelope is only available when you create or open a Composed drawing from a 3D task.

1. Right-click a Composed drawing document in the **Detail View** and select **Edit**.

The drawing opens in **SmartSketch Drawing Editor**.

Click **Detail Envelope** on the toolbar.

The Place Detail Envelope ribbon bar displays.

- 3. In the drawing document, select a drawing view. Zoom in on the area of the main drawing view to the location you want to place the detail envelope geometry.
- 4. Click Circle ⊕ or Polygon △ so you can begin defining the detail envelope.
- 5. Type the text for the detail view name in **Reference mark**. If a second line of text is needed, type the text in **Additional callout text**.
- 6. For a **Circle** ⊕ detail envelope, click at the center of the circle, drag for the radius, and click again to set the radius.
- 7. For a **Polygon** \triangle detail envelope, click points as needed to create the detail envelope geometry. Complete the polygon by moving the cursor over the starting point until the **Close Polygon** \nearrow symbol displays, then click to close the polygon.
- 8. You can now use this detail envelope to place a detail view in the drawing.

Place a Detail View

The following steps describe the basic workflow for placing a detail view based on a detail envelope.

1. Right-click a drawing document in the **Detail View** and select **Edit**.

The drawing opens in **SmartSketch Drawing Editor**.

- 2. In the drawing document, select a detail envelope to place a detail view. Select from the drawing area or in the Drawing View Explorer.
- 3. Click Place Detail View
 on the toolbar.

The Place Detail View ribbon bar displays.

- On the Place Detail View ribbon, make sure the settings are appropriate for your new detail view:
 - **Sheet Name** is disabled. By default, the view must be on the same sheet as the detail envelope.
 - Specify the View Style used for the view contents. Select More in the View Style list to view more style options.
 - Set the View Scale as needed. By default, the scale for the detail view is Fit to Scale.
- 5. Drag the preview outline of the view in the drawing area to the needed location. Click to place the view.

An outline of the view is placed. For marine mode, the view is also out-of-date in the Drawing View Explorer.

6. In plant mode, right-click the view in the drawing area and select **Update View**.

-OR-

In marine mode, right-click the view in the Drawing View Explorer and select Full Update.

Geometry, labels, and other annotations as defined by the view style display in the view.

7. Crop the view as needed by selecting the view and dragging the handles on the sides of the view.

Delete a Detail Envelope from a Drawing

- 1. Right-click a drawing document in the **Detail View** and select **Edit**.
 - The drawing opens in SmartSketch Drawing Editor.
- 2. If a detail view has not been created, right-click a detail envelope in the Drawing View Explorer and select **Delete**.
- 3. If a detail view has been created, right-click the detail envelope and select **Delete**.
 - The **Convert or Delete** dialog box displays.
- 4. To convert the existing view(s) associated to the detail envelope to an independent view(s), select the Convert to independent drawing view(s) option and click OK. To delete the existing drawing view(s) associated to the detail envelope, select the **Delete** option and click OK.

Delete a Detail View from a Drawing (Marine mode only)

- 1. Right-click a drawing document in the **Detail View** and select **Edit**.
 - The drawing opens in SmartSketch Drawing Editor.
- 2. In the Drawing View Explorer, right-click a view assigned to a drawing and select **Delete**.
- 3. Select **Delete** to delete the view permanently, or select **Unassign** to move the view in the Unassigned Folder.

■ NOTES

- You can either move a view created by rules to the **Unassigned Folder** or permanently delete a view.
- If you delete a view that is a parent of other views, such as a detail or section view, the Convert or Delete dialog box displays. Select Convert to independent drawing view(s) to save the child view as an independent drawing view, or select Delete to delete the child view. The view is deleted.
- When a view is out-of-date, the view icon looks like this:



- When a view is up-to-date, the view icon looks like this: Z.
- When a view is unassigned and in the Unassigned Folder of Drawing View Explorer, the view icon looks like this:

2D/3D Selection Command

Allows you to move back and forth between the SmartSketch Drawing Editor application window and the 3D window. This command is only available in SmartSketch Drawing Editor when you create a new drawing or open an existing drawing from a 3D task. This command is turned off by default when you open a document in the SmartSketch Drawing Editor

Behavior while Turned On (Object Select/Highlight Mode)

When the command is on, you can select geometry in a drawing view, and the software highlights and selects the corresponding object in the 3D model. The object is selected even if it is not currently visible or displayed in the 3D workspace. If you select an object in the 3D task window, the software highlights the corresponding geometry in the drawing view in SmartSketch Drawing Editor. However, if the object is not included in the drawing view, no geometry highlights. The filters in the drawing view style dictate the content of the drawing view.

You can also multi-select 2D objects (groups and views) using the CTRL key.

NOTE If you select a drawing view (instead of geometry in the view), the 2D/3D Selection command turns off automatically and the Associate Objects to View command is active. You can then modify the volumes or views associated to the selected view. For more information, see Associate Objects to View Command (on page 300).

Behavior while Turned Off (View Input Mode)

The command is off by default when you open a drawing.

When this command is off, geometry selected in a view is not actively linked to its 3D model object and the drawing document is in *view input* mode. You can modify the layout and contents of the drawing sheet and the properties of the drawing views.

▶ NOTE When 2D/3D Selection is off, you can click Associate Objects to View then move back and forth between the 3D application and SmartSketch Drawing Editor to associate drawing views to volumes in the model. For more information, see Associate Objects to View Command (on page 300).

Considerations for Using the 2D/3D Selection Command

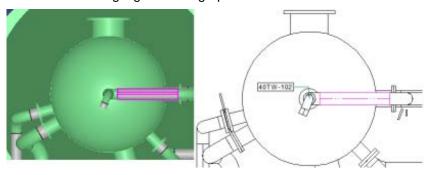
- **2D/3D Selection** , by default, only allows selection of 2D group elements that correspond to 3D model objects.
- You can use the spacebar key to toggle the selected drawing view when this command is used to associate objects to objects in the model. For more information, see Associate Objects to View Command (on page 300).
- Selecting a smart label in a SmartSketch Drawing Editor document selects the associated
 3D object in the 3D application.
- Selecting a volume in the 3D application selects the associated views in the SmartSketch Drawing Editor document.
- The 2D select set is cleared when you close the drawing document.

Compare 2D Drawing Object to 3D Model Object

The following steps show you how to compare drawing objects to 3D model objects.

- 1. In a 3D modeling task, such as Common, select **Tools > Drawings Console**.
- Right-click a drawing document and select Edit to open a drawing document in SmartSketch Drawing Editor.
- 3. In **SmartSketch Drawing Editor**, zoom into a drawing view to select the drawing object you want to compare to the 3D model.

4. Click **2D/3D Selection** to move focus to the 3D application. The 3D model object selects and highlights in the graphic windows.



Considerations for Using the 2D/3D Selection Command

The **2D/3D Selection** command, by default, only allows selection of 2D group elements that correspond to 3D model objects.

- You can use the spacebar key to toggle the selected drawing view when this command is used to associate objects to objects in the model.
- Selecting a SmartLabel in SmartSketch Drawing Editor document selects the associated 3D object in the 3D application.
- Selecting a volume in the 3D application selects the associated views in the SmartSketch Drawing Editor document.
- The 2D select set is cleared when you close the drawing document.

For more information on the commands available in **SmartSketch Drawing Editor**, see the *SmartSketch Drawing Editor Help*. You can also refer to the *Common User's Guide* for information on the **Tools > Drawings Console** command.

Using Scaled Sketching

In drawings, you can have multiple embedded views on a drawing sheet that are at different model scales (e.g. 1/4" = 1' and 1/8" = 1'). You can add graphics to the views and treat them as if they were actual model graphics. Use the Scaled Sketching command to draw all graphics at a consistent scale factor. The scale factor used is based on a user-selected SmartFrame.

For more information, see *Dimensioning Drawing Elements* in the *SmartSketch Drawing Editor User's Guide*.

See Also

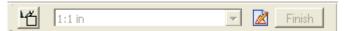
Scaled Sketching Command (on page 365) Scaled Sketching Ribbon (on page 365) Draw or Edit Objects at the Same Scale Factor (on page 363) Move a View with Scaled Sketching Objects (on page 365)

Draw or Edit Objects at the Same Scale Factor

Follow the steps below to use the Scaled Sketching command to draw or edit objects at a specific scale factor.

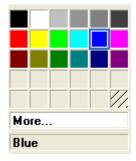
■ NOTE Make sure you have added the Scaled Sketching command to your Smart 3D toolbar. See Add the Scaled Sketching Button to the Smart 3D Toolbar.

1. Click **Scaled Sketching** to open the **Scaled Sketching** ribbon.



Scaled Sketching Ribbon (on page 365)

2. To set the color of any objects not being edited using the scale factor, click **Select color for referenced data** . Click on the required color from the color palette.



 Insert a SmartFrame (see Working with Object, Linking and Embedding) and set the scale factor of the SmartFrame, or click on an existing SmartFrame. The scale is read and taken from that SmartFrame. The selected scale type is displayed in the **Scaled Sketching** ribbon.

★ IMPORTANT

- Do not create or edit any layers when using the scaled sketching command.
- To get accurate dimensional data about scaled objects, be sure to dimension them
 while in scaled sketching mode. The dimensions are automatically configured to display
 at the value in which they were drawn.
- 4. Click Enter Scaled Sketch mode M to start drawing at the selected scale.

The system scales the drawing sheet to match the specified scale value.

- 5. Use the application's drawing/editing commands to draw the required graphics.
- 6. Click Finish to end the selected scale mode.

■ NOTES

- The only way to exit scaled sketching is to click Finish. All modifications you made in the document are retained.
- Whenever you finish using scaled sketching, the undo history is cleared and you cannot undo any previous edits. **Undo** is available after you continue working in SmartSketch Drawing Editor.
- Leaders must be placed on objects (Labels, Text Boxes, and so forth) that were placed during Scaled Sketch mode. If the objects were not placed during Scaled Sketch mode, the leader is not remembered in the SmartFrame after the view is updated.

Scaled Sketching Command

Enables you to edit or draw new objects at a scale that is different from the drawing scale for the active sheet. You can select a SmartFrame in the current document. The system then uses the scale factor of that SmartFrame.

See Also

Draw or Edit Objects at the Same Scale Factor (on page 363) Scaled Sketching Ribbon (on page 365)

Scaled Sketching Ribbon

The Scaled Sketching ribbon displays when you click **Scaled Sketching** on the Smart 3D **Drawings Compose** toolbar.

Select color for referenced data displays a color palette for you to set the color of any objects not being edited using the scale factor.



Displays the scale taken from the selected SmartFrame.

Enter Scaled Sketch Mode

Starts the system in using the scale factor for objects being drawn or edited.

Finish

Ends the scale mode.

See Also

Scaled Sketching Command (on page 365)
Draw or Edit Objects at the Same Scale Factor (on page 363)

Move a View with Scaled Sketching Objects

Follow the steps below to move a View that has objects drawn with the Scaled Sketching command.

- 1. If you are using the Scaled Sketching command, exit the command.
 - NOTE You cannot move a view while using the Scaled Sketching command.
- 2. Press **ALT** and click on the view you want to move.
- Release ALT.
- 4. Move the view to a new location.

When you press and release **ALT** while clicking on the view, all of the Scaled Sketching objects move with the view.

Draw a Grate Opening with Scaled Sketching

Follow the steps below to use the **Scaled Sketching** command to draw a grate opening in a slab at a specific scale factor.

1. From the Drawings environment, open a drawing. In this example, the drawing contains equipment.

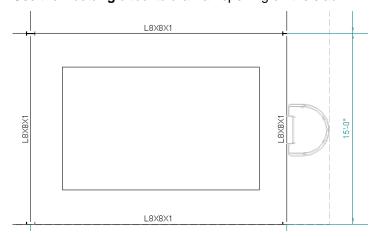
- 2. Click Scaled Sketching to open the Scaled Sketching ribbon.
- Click Select color for referenced data to set the color of any objects not being edited using the scale factor.
- 4. Click the SmartFrame in which you would like to sketch. The scale automatically matches the SmartFrame scale.

★ IMPORTANT

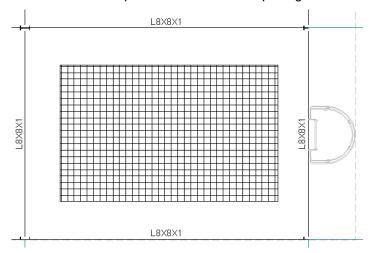
- Do not create or edit any layers when using the scaled sketching command.
- To get accurate dimensional data about scaled objects, be sure to dimension them while in scaled sketching mode. The dimensions are automatically configured to display at the value in which they were drawn.
- 5. Click **Enter Scaled Sketch mode** M to start drawing at the selected scale.

The system scales the drawing sheet to match the specified scale value.

6. Use the Rectangle tool to draw an opening on the slab.



7. Use the **Fill** tool to place a **Grid** fill in the opening.



8. Click Finish to end the selected scale mode.

■ NOTES

- Any objects drawn while in Scaled Sketching mode will move with the view if it is moved.
- The only way to exit scaled sketching is to click Finish. All modifications you made in the document are retained.
- Whenever you finish using scaled sketching, the undo history is cleared and you cannot undo any previous edits. **Undo** is available after you continue working in SmartSketch Drawing Editor.

Draw a Centerline with Scaled Sketching

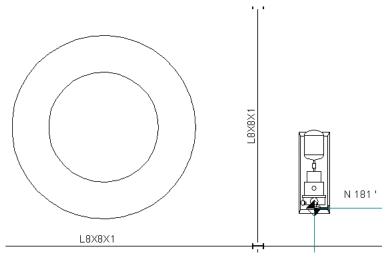
Follow the steps below to use the Scaled Sketching command to draw a centerline on an equipment object at a specific scale factor.

- 1. From the Drawings environment, open a drawing. In this example, the drawing contains equipment.
- 2. Click Scaled Sketching to open the Scaled Sketching ribbon.
- 3. Click **Select color for referenced data** to set the color of any objects not being edited using the scale factor.
- 4. Click the SmartFrame in which you would like to sketch. The scale automatically matches the SmartFrame scale.

★ IMPORTANT

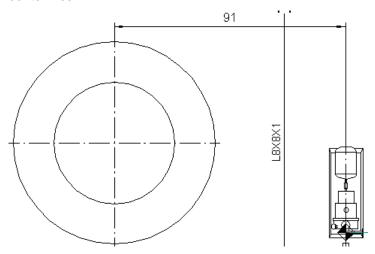
- Do not create or edit any layers when using the scaled sketching command.
- To get accurate dimensional data about scaled objects, be sure to dimension them while in scaled sketching mode. The dimensions are automatically configured to display at the value in which they were drawn.
- 5. Click Enter Scaled Sketch mode do to start drawing at the selected scale.

The system scales the drawing sheet to match the specified scale value.



6. Use the **Line** tool to draw a centerline on the equipment.

Use the **Distance Between** tool to place a dimension between the two equipment centerlines.



8. Click Finish to end the selected scale mode.

■ NOTES

- Any objects drawn while in Scaled Sketching mode will move with the view if it is moved.
- The only way to exit scaled sketching is to click Finish. All modifications you made in the document are retained.
- Whenever you finish using scaled sketching, the undo history is cleared and you cannot undo any previous edits. **Undo** is available after you continue working in SmartSketch Drawing Editor.

Draw a Textured Fill with Scaled Sketching

Follow the steps below to use the Scaled Sketching command to draw a grate opening in a slab at a specific scale factor.

- 1. From the Drawings environment, open a drawing. In this example, the drawing contains equipment.
- 2. Click Scaled Sketching to open the Scaled Sketching ribbon.
- 3. Click **Select color for referenced data** to set the color of any objects not being edited using the scale factor.
- 4. Click the SmartFrame in which you would like to sketch. The scale automatically matches the SmartFrame scale.

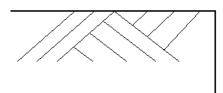
★ IMPORTANT

- Do not create or edit any layers when using the scaled sketching command.
- To get accurate dimensional data about scaled objects, be sure to dimension them
 while in scaled sketching mode. The dimensions are automatically configured to display
 at the value in which they were drawn.
- 5. Click Enter Scaled Sketch mode M to start drawing at the selected scale.

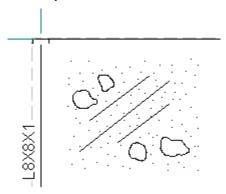
The system scales the drawing sheet to match the specified scale value.

6. Use the **Fill** tool to place an **Earth** fill texture inside a shape.

Earth



TIP You can also draw your own fill using **Scaled Sketching**. Below is an example of a manually drawn concrete fill.



7. Click Finish to end the selected scale mode.

■ NOTES

- Any objects drawn while in Scaled Sketching mode will move with the view if it is moved.
- The only way to exit scaled sketching is to click Finish. All modifications you made in the document are retained.
- Whenever you finish using scaled sketching, the undo history is cleared and you cannot undo any previous edits. **Undo** is available after you continue working in SmartSketch Drawing Editor.
- Fills can be placed while in scaled sketching mode if they are bounded by view objects, scaled sketch objects, or a combination of both.
- Fills are automatically deleted when you click **Finish** on the Scaled Sketching toolbar and:
 - Use scaled sketching for view A and place a fill on a scaled sketch object in view B.
 - Use scaled sketching for view A and place a fill on an object in view B.
 - Use scaled sketching for a view and place a fill on a non-scaled sketch, manually-drawn graphic.

Draw Rebar with Scaled Sketching

- 1. From the Drawings environment, open a drawing. In this example, the drawing contains equipment.
- 2. Click Scaled Sketching to open the Scaled Sketching ribbon.
- 3. Click **Select color for referenced data** to set the color of any objects not being edited using the scale factor.

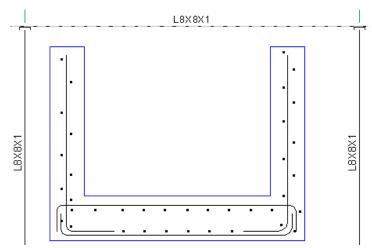
4. Click the SmartFrame in which you would like to sketch. The scale automatically matches the SmartFrame scale.

★IMPORTANT

- Do not create or edit any layers when using the scaled sketching command.
- To get accurate dimensional data about scaled objects, be sure to dimension them
 while in scaled sketching mode. The dimensions are automatically configured to display
 at the value in which they were drawn.
- 5. Click **Enter Scaled Sketch mode** Mode to start drawing at the selected scale.

The system scales the drawing sheet to match the specified scale value.

6. Use the tools in the **Draw** toolbar to draw rebar-reinforced concrete.



7. Click Finish to end the selected scale mode.

■ NOTES

- Any objects drawn while in Scaled Sketching mode will move with the view if it is moved.
- The only way to exit scaled sketching is to click Finish. All modifications you made in the document are retained.
- Whenever you finish using scaled sketching, the undo history is cleared and you cannot undo any previous edits. **Undo** is available after you continue working in SmartSketch Drawing Editor.

Draw an Opening in a Plate with Scaled Sketching

Follow the steps below to use the **Scaled Sketching** command to draw an opening in a plate at a specific scale factor.

- 1. From the Drawings environment, open a drawing. In this example, the drawing contains equipment.
- 2. Click Scaled Sketching to open the Scaled Sketching ribbon.
- 3. Click **Select color for referenced data** to set the color of any objects not being edited using the scale factor.

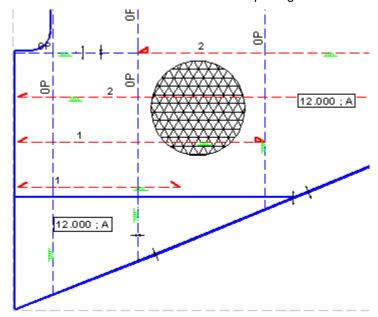
4. Click the SmartFrame in which you would like to sketch. The scale automatically matches the SmartFrame scale.

★IMPORTANT

- Do not create or edit any layers when using the scaled sketching command.
- To get accurate dimensional data about scaled objects, be sure to dimension them
 while in scaled sketching mode. The dimensions are automatically configured to display
 at the value in which they were drawn.
- 5. Click **Enter Scaled Sketch mode** Mode to start drawing at the selected scale.

The system scales the drawing sheet to match the specified scale value.

- 6. Use the **Circle** tool to draw an opening on the plate.
- 7. Use the Fill tool to add a Mesh fill to the opening.



8. Click **Finish** to end the selected scale mode.

■ NOTES

- For information on how to move a view that contains objects drawn using the Scaled Sketching command, see Move a View with Scaled Sketching Objects (on page 365).
- If a view is deleted and moved to the UnAssigned Folder, any sketches made in Scaled Sketching mode reside in the view. If the view is reassigned to a drawing, your sketches will appear in the view.
- The only way to exit scaled sketching is to click Finish. All modifications you made in the document are retained.
- Whenever you finish using scaled sketching, the undo history is cleared and you cannot undo any previous edits. **Undo** is available after you continue working in SmartSketch Drawing Editor.

See Also

Using Scaled Sketching (on page 363)

Retain Edits made inside a Drawing View

- 1. Double-click a drawing view's border to open the drawing view.
- 2. Make the required changes to the drawing view.
- 3. Click **File > Update** to save the changes.
- 4. Click File > Close.
- 5. Repeat the steps above as required for all other views.
- 6. Save and close the document.

The changes that are saved include:

- Layer-related changes (creating new layers, modifying a graphic's layer, and changing how layers are displayed using the Display Manager Dialog Box).
- Using the Drawing Editor to sketch new graphics, copy intelligent graphics, add text boxes, place fills, or place symbols.

When a drawing view is opened, the new window is maximized to fit the graphics within the view. Like the Scaled Sketching command, the sheet scale changes to match the view scale. If the view scale is not Fit to Scale, manual graphics drawn outside of the rectangle, enclosing the graphics within the view, are not visible after the next update.

■ NOTES

- Particular changes made to intelligent graphics affects what is retained when a drawing view is updated:
 - Copies of intelligent graphics are treated as manual graphics.
 - When no graphic rule is specified for a filter, or when the graphic rule does not specify a layer, intelligent graphics are placed onto the **Default** layer.
 - Intelligent graphics remain intelligent graphics after their layer is changed. If the corresponding 3D object is deleted, the layer change operation is lost.
 - It is recommended that you include any dimensions or leaders in the drawing sheet, instead of the drawing view, as dimensions and leaders connected to intelligent graphics are not retained when the view is updated.
 - Intelligent graphics that are deleted will reappear after the drawing view is updated.
 Select **Hide** to temporarily remove intelligent graphics.

Hide/Show Object Command

Hides or displays objects in the drawing view. You can hide certain objects from the drawing view by selecting object types from a list. You can also show objects that you have hidden using the same command.

When objects are hidden, they are added to a different drawing layer that is created automatically. When the objects are shown, they are restored to their original drawing layer and all annotations are restored.

■ NOTE You can also hide/show symbol objects that have Replace Object(s) with Symbol rule applied on them.

See Also

Hide/Show Object Ribbon (on page 373) Hide an Object (on page 373) Show an Object (on page 374) Copy an Object (on page 374)



Subtracts objects from the view.



Adds objects to the view.

View Name

Displays the selected view name.

Hide Options

Displays the list of hide options available in the selected drawing view. The default option is **All**.

- All Hides graphics, labels and dimensions.
- Graphics only Hides only graphics.
- Graphic/Labels Hides graphics and corresponding labels.
- Graphic/Dimensions Hides graphics and related dimensions.

Object Filter

Displays a list of object types in the selected drawing view. The default option is **All**. When you select an object in the drawing, the **Object Filter** displays associated object type automatically.

Copy Graphics

Copies only graphics into the smart frame. Ignores any applied hide options. By default it is disabled. Click **Hide** to enable this command.

Finish

Saves and exits the **Hide/Show** command. Click on the drawing view to enable this command.

NOTE You must update the drawing after hiding or showing objects.

Hide an Object

1. In the **Draw** toolbar, click **Hide/Show** 🔂.

The **Hide/Show** ribbon appears in the toolbar area.

NOTE The Hide The mode is selected by default.

2. Select a drawing view.

Hide disabled.

- 3. Select an hide option from the Hide Options list.
- Select an object type from the Object Filter list.
- 5. Select the objects you want to hide.

As you select the objects, the **Finish** command is enabled, and the selected objects are highlighted in the drawing view depending on the hide option. For example, if the hide option is **Graphics/Labels**, the selected graphic and related labels are highlighted in the drawing view.

- 6. Click Finish to save your changes.
 - NOTE You must update the drawing after hiding objects.

Show an Object

- 1. In the **Draw** toolbar, click **Hide/Show** .

 The **Hide/Show** ribbon appears in the toolbar area.
- 2. Click Show 🛂
- 3. Select a drawing view.

All hidden objects in the selected view are highlighted. Except for **Show** the remaining options on the ribbon bar are disabled and set to their respective default values.

4. Select the objects you want to hide. Only highlighted objects are displayed in the drawing view.

The Finish command is enabled.

- 5. Click Finish to save your changes.
 - NOTE You must update the drawing after showing objects.

Copy an object

1. In the **Draw** toolbar, click **Hide/Show 1**.

The Hide/Show ribbon appears in the toolbar area.

- NOTE The Hide amode is selected by default.
- 2. Select a drawing view.
- 3. Click Copy Graphics 🗗.
- 4. Select an object to copy.
- 5. Click Finish to save your changes.

You can see the copied object with no labels.

■ NOTE To view the copied object, double-click the drawing view. Move the copied object, and then turn on the hidden layer. For more information on layer display, see *Layers Command* in the *SmartSketch Drawing Editor User's Guide*.

Copy and Paste View Command

- © Copies an orthographic drawing view and places the copy on the same sheet.
- 1. Select an orthographic drawing view, and then click Copy and Paste View 🗎.
- 2. On the target drawing sheet, click **Paste** ■. *The view displays*.
- 3. Drag the drawing view to a new location on the sheet.
- TIPS You must use Copy and Paste View are rather than the traditional Cut, Copy, and Paste commands on views if you want to maintain the associations.
- If you try to drag and drop a view using a CTRL key operation, or if you try to cut a view, Smart 3D displays a message asking you to use **Copy and Paste View** instead.
- If you try to copy a view, the software displays a message telling you that the copied view will be an unassociated view. You then have the choice to continue or not.

Move View Command

Moves one or more views from a composed drawing document to another composed drawing document. The new drawing can be in the same component or in a different component.

A moved section view keeps its association with its cutting plane in the parent view, even when the parent view is in a different drawing. A moved detail view keeps its association with its envelope in the parent view, even when the parent view is in a different drawing.

Report and key plan views are moved automatically if the parent view is also moved. Dependent report and key plan views cannot be moved if the parent view is not moved.

Views cannot be moved to a drawing that is being edited or to a drawing to which you do not have write permissions.

■ NOTES

- Views cannot be moved in orthographic drawings by query or volume (spatial) drawings.

Move View Dialog Box (on page 376)

What do you want to do?

- Move a view to a different drawing (on page 375)
- Move multiple views to a different drawing (on page 376)

Move a view to a different drawing

- 1. Click Move View 34.
- 2. Select a view in the drawing area.

NOTE Alternatively, select the view first, and then click Move View 34.

The Move View dialog box appears.

- Expand the appropriate folder and component, and select the destination drawing document.
- 4. Click OK.

The selected view is moved to the destination drawing.

Move multiple views to a different drawing

- 1. Select two or more views in the drawing area.
- 2. Click Move View 34.

The Move View dialog box appears.

- Expand the appropriate folder and component, and select the destination drawing document.
- 4. Click OK.

The selected views are moved to the destination drawing.

Move View Dialog Box

Displays the Drawings and Reports tree view of folders, components, and drawings in the model. The tree view displays only drawings to which you have write permissions. Expand the appropriate folder and component, and then select the drawing to which you want to move your selected views.

OK

Closes the dialog box and moves the view to the selected drawing document.

Cancel

Closes the dialog box without moving the view.

■ NOTE For drawings by rule, you can only move the selected view to drawings and drawing sheets under the same component as the selected view. Only the drawings and sheets in that component are displayed.

Highlight Annotations Command

Highlights labels, dimensions, and customized graphics based on the options that you select. The options display on the left side of the drawing window.

This command is useful when you are troubleshooting labels and dimensions, such as when you update a drawing and are looking for certain label or dimension types.

Highlight Dialog Box (on page 377)

Highlight Dialog Box

Allows you to specify label and dimensions that you want to see highlighted on a drawing. This dialog box appears docked on the left side of the drawing by default. You can undock it if needed.

To see all of the options, use the scroll bar on the right side of the dialog box.

Highlight

Click this button to highlight the specified items in the drawing.

Clear

Clears all highlighted items on the drawing and removes all the items in the select set.

Clear Options

Clears all options on this dialog box.

Close

Ends this command.

Add to Select Set

Adds the highlighted items to the set of all selected items.

Choose Highlight Color



Specifies the color of the selected items on the drawing.

Labels

Unmodified System Placed

Highlights labels that have not been modified on the drawing and that were placed by the software.

User Placed

Highlights labels that a user placed on the drawing.

Modified System Placed

Highlights labels that have been modified on the drawing and that were placed by the software.

Deleted

Highlights labels that have been deleted.

Unassociated

Highlights manual labels that are no longer associated to at least one of their original geometry elements. Loss of association can occur after a view update when the **Default Graphics** or **Drawable Default Graphics** custom graphic modules are used in a view style. For more information, see *Default Graphics* and *Drawable Default Graphics* in the *Drawings and Reports Reference Data Guide*.

Additional Label Filters

Labels By Name

Click this option in order to choose a specific label type. When you check this box, the dropdown list is enabled. You can choose **More** in the dropdown list to view all label types in the current drawing.

Corrupt

Highlights labels that have a problem, such as not being connected to the correct object.

Include

Include Related Leaders

Highlights leaders that are related to the specified labels.

Include Other Relations

Highlights cutting planes or detail envelopes that are related to the specified labels.

Include Related Lines

Highlights lines that are related to the specified labels.

Dimensions

Unmodified System Placed

Highlights dimensions that have not been modified and that were placed by the software.

User Placed

Highlights dimensions that a user placed on the drawing.

Modified System Placed

Highlights dimensions that have been modified and that were placed by the software.

Deleted

Highlights dimensions that have been deleted.

Paper To Model

Highlights dimensions from an object drawn on the paper to an object that is in the model.

Unassociated

Highlights manual dimensions that are no longer associated to at least one of their original geometry elements. Loss of association can occur after a view update when the **Default Graphics** or **Drawable Default Graphics** custom graphic modules are used in a view style. For more information, see *Default Graphics* and *Drawable Default Graphics* in the *Drawings and Reports Reference Data Guide*.

Additional Dimension Filters

Dimensions By Name

Click this option in order to choose a specific dimension type. When you check this box, the dropdown list is enabled. You can choose **More** in the dropdown list to view all dimension types in the current drawing.

User Graphics

User Graphics

Highlights items, such as lines, that a user placed on the drawing.

User Scaled Dimensions

Highlights dimensions on the items that a user placed on the drawing.

Leaders

Disconnected Leaders

Highlights leaders that are disconnected from labels on the drawing.

See Also

Highlight Annotations Command (on page 376)

Clear Manual Edits Command

Permanently clears all manual edits made to labels and dimensions in the selected views.

You must first select one or more views before clicking this command.

Associate Graphics to Graphic View Command

Associates and disassociates manually drawn objects, such as lines, circles, symbols, and text boxes, to graphic views. You can use **Associate Graphics to Graphic View** in composed drawings and Drawings by Rule components. You cannot associate objects with a report view or a pasted view, or if the objects are already associated with another view. To associate the objects to another view, you must first dissociate the objects.

■ NOTES

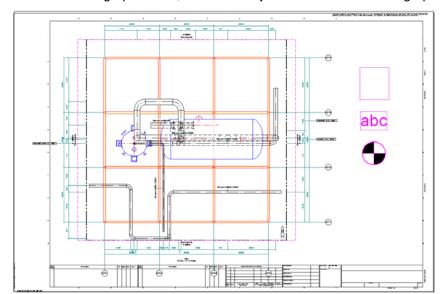
- You can use Move View it to move the view and the associated objects.
- To view all objects that are associated with a graphic view, press ALT and select the graphic view.

What do you want to do?

- Associate Graphics to Graphic View (on page 379)
- Disassociate Graphics from Graphic View (on page 380)

Associate Graphics to Graphic View

1. Select a graphic view.

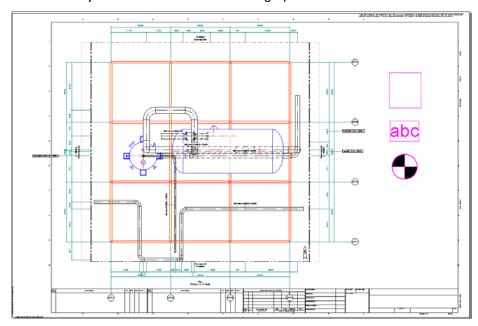


2. In the selected graphic view, select the objects to associate to the graphic view.

- 3. Click **Associate Graphics to Graphic View** ... *A message displays the number of associated objects.*
- 4. Click OK.

Disassociate Graphics from Graphic View

1. Select the objects to disassociate from the graphic view.



2. Click Associate Graphics to Graphic Views ...

A warning message displays.

3. Click Yes to remove all associated inputs from each object.

A message displays the number of disassociated objects.

4. Click OK.

Template Toolbar

This toolbar is available when you click **Edit Template** on a component shortcut menu.

Place Drawing View Command (Template Toolbar)

Places a two-dimensional drawing view for 3D volume drawings. This command is available when you use the **Edit Template** command available on the shortcut menu for a volume drawing component in the Drawings and Reports task.

When you place drawing views, the software automatically saves the views to the **DwgTemplate** layer when you save the document.

See Also

Manual Place View Ribbon (on page 385)
Drawing View Properties Dialog Box (Drawing View Shortcut Menu) (on page 382)

Place a Drawing View for Volume Drawings

Before placing a drawing view for a volume drawing, you must have a View Style. For more information, see *Define View Style Command (Tools Menu)* in the *Drawings and Reports Reference Data Guide,* accessible using the **Help > Printable Guides** command in the Drawings and Reports task.

- 1. Right-click a volume drawing component in the **Management Console**, and then click **Edit Template** on the shortcut menu.
- 2. Select a template on the **Select Template** dialog box and click **OK**. The drawing template opens in **SmartSketch Drawing Editor**.
- 3. Click Place Drawing View
 on the toolbar.
- 4. Click a point on the drawing sheet to define the first corner of a rectangle, and hold the mouse button down while dragging the mouse diagonally. Release the button at a second point.
 - TIP You can use **PinPoint** (or or both) to place views precisely.



The drawing view highlights on the drawing and the **Drawing View Properties** dialog box displays.

5. On the **Drawing View Properties** dialog box, define the necessary information on the **View** tab in the **Name** and **Description** boxes.

- 6. In the **Style** box, select a view style.
- 7. In the **Orientation** box, select an orientation such as **Looking Plan**.
- 8. In the **Scale** box, select a scale, or choose **Custom** and define the values in the boxes to the right.

TIPS

- If you choose **Custom**, you must type values that are greater than zero in the boxes at the right.
- The software converts the values that you type to the default unit of measure. For example, if you type values of 1 in to 1 ft, the software may convert those values to mm.
- 9. Specify additional information as necessary on the Format tab.
- 10. Click **OK**.
- 11. Click File > Save and exit SmartSketch Drawing Editor.

■ NOTES

- To associate volumes with the view, switch to the Space Management task and click one of the Place Drawing Volume commands.
- You can resize a drawing view by dragging its handles.
- You cannot undo a drawing view delete operation. A message box displays when you press
 Delete for a selected view, providing a chance to cancel the operation.

Drawing View Properties Dialog Box (Drawing View Shortcut Menu)

Sets drawing view properties for a 3D volume drawing. This dialog box appears when you select a drawing view in a volume drawing then right-click and select **Properties** on the shortcut menu.

See Also

Place Drawing View Command (Template Toolbar) (on page 381) Info Tab (Drawing View Properties Dialog Box) (on page 382) Format Tab (Drawing View Properties Dialog Box) (on page 383) View Tab (Drawing View Properties Dialog Box) (on page 383)

Info Tab (Drawing View Properties Dialog Box)

Provides information about the frame around a drawing view. This information is view-only. You cannot make changes.

Type

Displays the category of the selected element.

Sheet

Displays the name of the drawing sheet that contains the selected element.

Layer

Shows the layer that contains the selected element.

Origin

Specifies the coordinates, or location, of an element along the X- and Y-axes.

See Also

Place Drawing View Command (Template Toolbar) (on page 381)
Drawing View Properties Dialog Box (Drawing View Shortcut Menu) (on page 382)

Format Tab (Drawing View Properties Dialog Box)

Formats the frame around a drawing view.

Show Border

Displays the frame around the object.

Color

Sets the color of the frame.

Line Width

Sets the line thickness on the frame.

Line Type

Overrides a line type for a drawing sheet or embedded object and sets another line style for an element or a linked object.

See Also

Place Drawing View Command (Template Toolbar) (on page 381)
Drawing View Properties Dialog Box (Drawing View Shortcut Menu) (on page 382)

View Tab (Drawing View Properties Dialog Box)

Sets the drawing view style and other properties for a selected drawing view in a 3D volume drawing.

▶ NOTE If the drawing document is a 3D composed drawing, refer to *Drawing View Properties Dialog Box (Place View Command) - Steel Order Drawings* (on page 281) for information on the properties shown on this tab.

Name

Specifies a name for the view. You must type a name in order to create a view.

Description

Describes the content of the view. This description is optional.

Style

Specifies a view style, which includes rules for filters, updates, and graphics. The view style controls the output characteristics of the view on the generated drawing. The list displays the 10 most recently used view styles in the session. Click **More...** to display the **Select View Style** dialog box.

Orientation

Positions the view in the model. For example, the view can look north, south, east, or west in the model.

Use object coordinate system

Specifies that the drawing takes its viewing direction directly from the object within the drawing view, not from the orientation of the drawing view.

Scale

Sizes the view as a ratio of drawing size to actual model size. If you select **Custom** for the scale, you must type values that are greater than zero in the boxes at the right to set the scale-to-scale ratio for object-to-drawing view.

■ NOTES

- For **Custom** scale, the default is to the document's unit of measure setting. For example, if you define values of **1** in to **1** ft, the values are converted to **mm** if that is the default unit of measure.
- Do not use negative values when defining custom scale values.
- For volume drawings, if the volume is *too big* for the drawing view, the software centers the volume in the drawing view.

Navigation rule

Specifies the navigation rule to use to traverse elements to be included in the range for Orthographic Drawings by Query. This dropdown only appears when you are placing views on drawings created by the Orthographic Drawings by Query component in the Drawings and Reports 3D task. The navigation rule can also optionally return separate object collections, whose range is included in the 3D object range. If no navigation rule is specified, the drawing object collection includes everything in the 3D object range. The delivered rules are:

- HngSupSimpleNavigator.dll Specific to Hanger and Support objects. Returns the HangersSupport objects, support components, supporting objects, supported objects, and the design children (recursively). It also returns the control points on the HangerSupport objects and support components.
- HngSupRangeNavigator.dll Same as HngSupSimpleNavigator.dll with design children collection to extent the HangerSupport range.
- DrawingSpoolNavigator.dll Specific to Spools. Returns the Spool, its connected parts, and their features.
- AssemblyNavigator.dll Specific to Assemblies. Returns the assemblies, pipe spool, penetration spool, its connected parts, and their features.

Convert report output to text boxes (no Excel)

Specifies that any report associated with this drawing view will be converted automatically to native text box format, even if the report is an Excel spreadsheet report. For information on converting Excel spreadsheet reports, see *Convert Excel Spreadsheet Reports to Native Text Box Format Custom Command* (on page 403).

Flush Threshold

Sets a parameter for memory management. When the number of objects processed for a drawing document reaches the **Threshold** value, they are removed from memory. If they are needed later, they are recalled from the database. This property helps improve drawing update performance. The **Flush Threshold** property is only available for composed drawing documents. The default value is **2000**, with a range of **5-5000**. The higher number is faster but uses more memory, which is acceptable for smaller drawings. Lower numbers are

slower but allow larger drawings to complete faster.

NOTE If a drawing document does not successfully update in the Drawings and Reports 3D task, check the error log for the drawing document. If the error log shows memory overflow errors, lower the **Flush Threshold** value.

VHL Precision

Sets a parameter for Hidden Line removal processing to consider two lines as identical. This property setting has a direct impact on the visibility of the lines in the drawing. It improves drawing update performance, but it may reduce drawing quality. The **VHL precision** property is only available for composed drawing documents. The default value is **0.000001**, with a range of **0.001 to 0.000001**. The smaller the number, the more accurate the graphic elements are in the 2D view.

NOTE If some intersections of complex surfaces appear on/off along the curve, the precision of the VHL may be too restrictive compared to the precision by which the surfaces were created. Lowering the **VHL Precision** value may help the display of the intersections, but lowering the value too much could degrade the overall quality and the visibility of the drawing details.

Geometry Validation

Sets a parameter for drawing completion and quality to improve drawing update performance. The **Geometry Validation** property is available for composed drawing documents only. The default value is **Off**. When set to **Off**, the drawing document completes, but invalid geometries are left out. If set to **On**, the drawing document does not complete if invalid geometries are encountered during update.

Tag

Identifies the reference mark for the selected detail view. This property is only available when a detail or section view is selected.

Caption

Identifies the callout text assigned to the selected detail view. This property is only available when a detail or section view is selected.

See Also

Place Drawing View Command (Template Toolbar) (on page 381) Drawing View Properties Dialog Box (Drawing View Shortcut Menu) (on page 382)

Manual Place View Ribbon

Sets options for placing a snapshot drawing view. These options include the view name, style, and scale.

View Name

Displays the name of the view you created in a 3D task.

View Style

Specifies a view style, which includes rules for filters, updates, and graphics. The view style controls the output characteristics of the view on the generated drawing. The list displays the 10 most recently used view styles in the session. Click **More...** to display the **Select View Style** dialog box.

View Scale

Specifies a scale. You can choose **No Scale** if the view is not to scale. Choose **Custom** if you want to specify your own scale and then type values in the two boxes at the right of the ribbon. If you choose **Custom** for the scale, you must type values that are greater than zero in the boxes at the right.

See Also

Place Drawing View Command (Template Toolbar) (on page 381)

Place Report Command (Template Toolbar)

Embeds a report in a drawing view on a 3D drawing. You must select a drawing view on the drawing before you can embed a report.

The report queries on the items in the drawing view.

When you place reports, the software automatically places them on the **DwgTemplate** layer so that they will be preserved when you update the drawing document.

■ NOTES

- For embedded reports to run on computers with Microsoft Office XP, you must modify the security settings in Excel to allow Visual Basic projects to run. To change this setting, open Excel, and click Tools > Macros > Security. On the Trusted Sources tab, select Trust access to Visual Basic Project. This setting must be modified before you update the drawing and generate the report within it.
- When defining embedded report layout (sizing of columns and rows), consider the report usage first. Because of a Microsoft limitation concerning the size of Windows metafile objects within other applications, the data displayed may be incomplete. Therefore, no column should be out of screen when using 100 percent zoom for the report. Otherwise some columns are ignored when the report is embedded within the drawing. The same limitation exists for rows. To preserve the maximum number of rows displayed, the total header row(s) height should be a minimum of the overall report. Using Microsoft Excel default settings, the maximum number of columns is approximately 20 and the maximum number of rows is approximately 75 (including header rows). For more information on setting the defaults in Microsoft Excel, see your Microsoft Excel documentation.

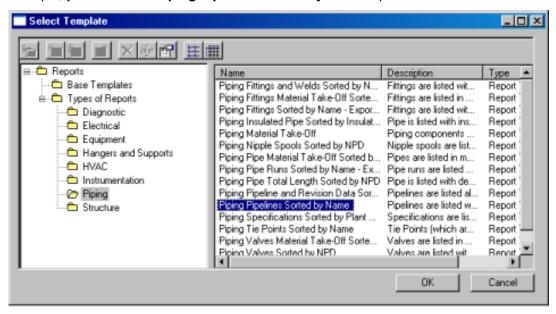
See Also

Report Properties Dialog Box (Place Report View Command) (on page 320)

Place an Embedded Report in a Volume Drawing

- 1. For volume drawings, right-click the volume drawing component in the **Drawing Console** and select **Edit Template**.
- 2. Select a template on the **Select Template** dialog box and click **OK**. The drawing template opens in **SmartSketch Drawing Editor**.
- 3. Click Place Report , and then click on the drawing view border.

4. On the **Select Template** dialog box, select a report template from the hierarchy. For example, you can select **Piping Pipelines Sorted by Name** report.



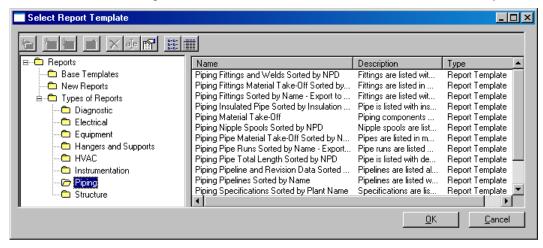
- 5. Click **OK** on the dialog box.
- 6. In **SmartSketch Drawing Editor**, click to place the report.

■ NOTES

- Only one report per drawing view is permitted.
- For more information about Microsoft Office and service packs, refer to the *Microsoft web site* (http://www.microsoft.com/) (http://www.microsoft.com/).
- If you are using Office 2003, in Microsoft Excel under Tools > Macro > Security > Trusted
 Publishers tab, check the Trust Access to Visual Basic Project option.
- If you are using Office 2007 and Office 2010, click the Microsoft Office button to access Excel Option. Go to the Trust Center category and select the Trust Center Settings button. Select the Macro Settings category and check Trust access to the VBA project object model.
- When defining embedded report layout (sizing of columns and rows), consider the report usage first. Because of a Microsoft limitation concerning the size of Windows metafile objects within other applications, the data displayed may be incomplete. Therefore, no column should be out of screen when using 100 percent zoom for the report. Otherwise some columns are ignored when the report is embedded within the drawing. The same limitation exists for rows. To preserve the maximum number of rows displayed, the total header row(s) height should be a minimum of the overall report. Using Microsoft Excel default settings, the maximum number of columns is approximately 20 and the maximum number of rows is approximately 75 (including header rows). For more information on setting the defaults in Microsoft Excel, see your Microsoft Excel documentation.

Select Report Template Dialog Box

Selects a report template. This dialog box appears when you click **Place Report** when editing a 3D drawing template. By browsing through the hierarchy, you can find any report template in the Catalog database. After you select a template, the software generates the report. You can resize the dialog box and the columns to view the information more clearly.



Properties

Displays the properties of the selected item. All properties on the **Properties** dialog box are read-only.

List View

Sets the dialog box to display items in a list view.

Grid View

Sets the dialog box to display items in a spreadsheet-style grid view.

■ NOTE The **Place Report** commands creates reports based on a selected report template. You can also select a report template and view its properties. The buttons that are grayed out are not available when using these commands.

Place Key Plan Command (Template Toolbar)

Places a key plan on a drawing template. The Place Key Plan command is enabled when you select a drawing view. The command displays the **Select Key Plan** dialog box to specify a key plan type to associate with the selected drawing view.

When you place key plan views, the software automatically places them on the **DwgTemplate** layer so that they will be preserved when you update the drawing document.

A key plan is a small graphical representation of a drawing volume you defined in the geographic area where engineering is taking place. One key plan may serve for an entire project. Multiple key plans may be established per discipline. One drawing view may have multiple key plans.

See Also

Select Key Plan Style Dialog Box (on page 389)

Select Key Plan Style Dialog Box

Specifies a style for the selected key plan on a volume- drawing template. Select a style from the **Rule Name** hierarchy.

See Also

Key Plan Properties Dialog Box (on page 389) Place Key Plan Command (Template Toolbar) (on page 388)

Key Plan Properties Dialog Box

Sets options for a key plan on a drawing template.

See Also

Place Key Plan Command (Template Toolbar) (on page 388) Info Tab (Key Plan Properties Dialog Box) (on page 389) Format Tab (Key Plan Properties Dialog Box) (on page 389) Key Plan Tab (Key Plan Properties Dialog Box) (on page 390)

Info Tab (Key Plan Properties Dialog Box)

Provides information about a key plan. This information is read-only. You cannot make changes.

Type

Displays the category of the selected element.

Sheet

Displays the name of the drawing sheet that contains the selected element.

Layer

Shows the layer that contains the selected element.

Origin

Specifies the coordinates, or location, of an element along the X- and Y-axes.

See Also

Key Plan Properties Dialog Box (on page 389) Place Key Plan Command (Template Toolbar) (on page 388)

Format Tab (Key Plan Properties Dialog Box)

Formats the frame around a key plan.

Show Border

Displays the frame around the object.

Color

Sets the color of the frame.

Line Width

Sets the line thickness on the frame.

Line Type

Overrides a line type for a drawing sheet or embedded object and sets another line style for an element or a linked object.

See Also

Place Key Plan Command (Template Toolbar) (on page 388) Key Plan Properties Dialog Box (on page 389)

Key Plan Tab (Key Plan Properties Dialog Box)

Sets the properties for a selected key plan.

Name

Specifies a unique name for the key plan.

Description

Describes the key plan contents.

Style

Indicates the key plan view style used. Select **More** to display the **Select Key Plan Style** dialog box. For more information, see *Select Key Plan Style Dialog Box* (on page 389).

Scale

Indicates the scale assigned to the key plan with regard to the associated drawing view. If you select **Custom** for the scale, you must type values that are greater than zero in the boxes at the right to set the scale-to-scale ratio for key plan-to-drawing view.

■ NOTES

- For Custom scale, the default is to the document's unit of measure setting. For example, if
 you define values of 1 in to 1 ft, the values are converted to mm if that is the default unit of
 measure.
- Do not use negative values when defining custom scale values.
- Orientation of the key plan graphic is specified as part of the key plan style.

See Also

Key Plan Properties Dialog Box (on page 389)
Place Key Plan Command (Template Toolbar) (on page 388)

Place a Key Plan

- 1. In the **Management Console**, right-click a volume drawings component, then click **Edit Template**.
- 2. If no template is defined for the volume drawings component, select a template in the **Select Template** dialog box, and click **OK**.
- 3. Select a drawing view on the template.
- 4. On the **Template** toolbar, click **Place Key Plan** 11.

5. Click two points to place the key plan view.



On the second click, the software displays the **Key Plan Properties** dialog box.

6. On the dialog box, specify the settings for the key plan.

■ NOTES

- Multiple key plans can be associated with a single drawing view.
- For Custom scale, the default is to the document's unit of measure setting. For example, if
 you type values of 1 in to 1 ft, the values are converted to mm if that is the default unit of
 measure.
- Do not use negative values when typing custom scale values.
- Orientation of the key plan graphic is specified as part of the key plan style. For more information, see "Define View Style Command (Tools Menu)" in the *Drawings and Reports* Reference Data Guide.

For more information, see Key Plan Drawing Requirements and Key Plan View Styles in the Drawings and Reports Reference Data Guide.

Edit Border Template Toolbar

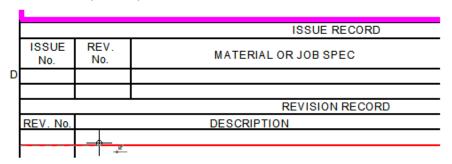
Available when you click **Tools > Edit Border Template** in the Drawings and Reports task.

Place Drawing Property Label Command (Drawing Labels Toolbar)

Positions drawing property labels in the title block of a template. The command allows you to place drawing properties defined in the 3D drawing XML schema file as title block information on the drawing. When you click **Place Drawing Property Label**, the **Place Drawing Property Label** ribbon appears in the toolbar area.

Place Drawing Property Label Ribbon (on page 392)

When placing drawing property labels, zoom into the border area where you want to place the label for more precise placement.



Place Drawing Property Label Ribbon

Sets options for drawing property label placement on a border template. You can access this ribbon when you click the **Place Drawing Property Label** command when editing a border template in SmartSketch Drawing Editor.

When you place drawing property labels, the software automatically makes the **DwgTemplate** layer active. The labels need to be on this layer so that they are preserved when you update the drawing.



You set options as needed within the **Place Drawing Property Label** ribbon, then place the label in the drawing border title block area.

Label Set

Specifies a property category. This list shows the categories of drawing properties available for the current drawing. The label set controls the fields listed in the Field dropdown list and the enabling of other options on the ribbon.

Fields

Lists the properties available in the selected Label Set. This is the information you are placing on the title block of the drawing.

Function

Provides positioning functions for the label. The options available are **Index**, **First**, and **Last**. This control works with the **Function Operator** and **Function Argument** fields to set the position of the label within the title block area.

Function Operator

Works with the **Function** and **Function Argument** fields to set the position of the label within the title block area. The value is controlled by the **Function** selection. This field is not editable.

Function Argument

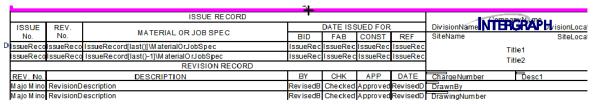
Sets a value to add or subtract from the **First** or **Last** settings in the **Function** field. This field works also works with the **Function Operator** field to set the position of the label within the title block area.

Alternative Text Value

Specifies alternative text to use if the selected property has no current value.

Display Label Names

Turns the label names on/off in the open border template:



A... More

Expands the ribbon to include additional formatting controls.



The controls on the expanded ribbon include:

Style

Sets the overall style used within the label.

Font

Sets the font for the label text.

Font Size

Sets the font size for the label text.

Textbox Width

Specifies the width of the text box.

Place a Drawing Property Label on a Template

If you want to place a custom attribute label on a template, see *Place a Custom Drawing Property Label on a Template* (on page 394).

1. Click **Tools > Edit Border Template**.

The Select Template dialog box displays.

2. Select a template, and click **OK**.

The template opens in SmartSketch Drawing Editor.

3. On the **Drawing Labels** toolbar, click **Place Drawing Property Label** ...

The Place Drawing Property Label ribbon displays.



TIP Click **Display Label Names** to show the labels names as they apply to the open border template:

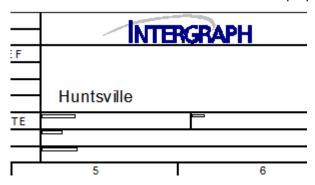


- 4. In the Label Set list (the first dropdown on the ribbon), select a label set (set of drawing properties). The list reflects the label sets within the drawing XML schema. The Label Set selection controls the contents of the Fields list and the enabling of other controls on the ribbon.
- 5. In the **Fields** list (the second dropdown on the ribbon), select a field to use as your title block label.
- 6. Specify a **Function**, setting the **Function Argument** if needed.
- 7. Provide alternative text for cases when the label property could be blank.

- 8. Click More to expand the ribbon and set formatting options. Select the Style, Font, Font Size, Text Color, Textbox Height, and Textbox Width.
- 9. Zoom into the area of the border where you want to place the drawing property label. Click the template to place the label.

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			ISSUE RECORD
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D			
	REVISION RECORD		
	REV. No.		DESCRIPTION
		- E	

10. Continue placing labels on the template as necessary. For example, the following border shows that the **General** label for the **Location** property has been placed as **Huntsville**.



- 11. Right-click to end the command.
- 12. Save the changes to the template before closing **SmartSketch Drawing Editor**.

See Also

Publishing Title Blocks (on page 425)

Place a Custom Drawing Property Label on a Template

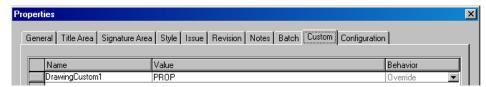
The following steps show how to add a custom attribute drawing property label to a drawing template. To add non-custom property labels to the template, see *Place a Drawing Property Label on a Template* (on page 393).

- Create a Custom Attribute workbook using Excel. This is the bulkload file for the custom attribute and names the attribute. For example, you could create a file called CustomAttributes.xls that contains the definition for a custom attribute. For more information on the format for the workbook, see Drawings-ExtendCustomAttributes Workbook in the Drawings and Reports Reference Data Guide.
- 2. Bulkload the Custom Attribute workbook. For more information on populating Excel workbooks and bulkloading, see the *Reference Data Guide* available from **Help > Printable Guides**.
- 3. Create an .xsd file and add a line that defines the attribute in the dropdown list when you edit a template and use the **Place Drawing Property Label** command. For example, using

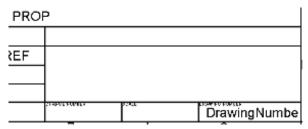
DrawingCustom1 as the attribute name, the line would be <xs:element name="DrawingCustom1">.

- **NOTE** You can only use letters, numbers, and underscores when defining the **element name**. The name must also begin with a letter.
- 4. Add a line that points to the **AttributeName** property of the bulkloaded attribute. For example, using the above attribute name, the necessary line would be *<pk* name="DrawingCustom1"/>. Your .xsd file would look similar to the following:

- **NOTE** The **pk name** in the Custom.xsd file must match the related **AttributeName** property of the bulkloaded attribute.
- 5. Make sure the .xsd file is in the \(\sigma\)ymbols\\Drawings\\Catalog\\Labels\\Border\\Schema\) folder.
- 6. In the Drawings and Reports task, click Tools > Edit Border Template.
- 7. On the **Select Template** dialog box, select a template, and click **OK**. The template opens in **SmartSketch Drawing Editor**.
- 8. In the Label Set list (the first dropdown on the ribbon), select Custom.
- 9. In the **Fields** list (the second dropdown on the ribbon), select the Custom attribute property to use as your title block label.
- 10. Zoom into the area of the border where you want to place the drawing property label. Click the template to place the label.
- 11. Continue placing labels on the template as necessary.
- 12. Right-click to end the command.
- 13. Save the changes to the template before closing **SmartSketch Drawing Editor**.
- 14. Update any drawing documents associated with the modified template. For more information, see *Updating Documents* (on page 78) in the *Drawings Help*.
- 15. Right-click the drawing and select **Properties**.
- 16. Go to the **Custom** tab. The new Custom attribute property is shown on the tab.



17. To verify the custom drawing property is added to the drawing title block, right-click the drawing and select **Open**. The custom drawing property includes the Custom Property attribute in the title block.



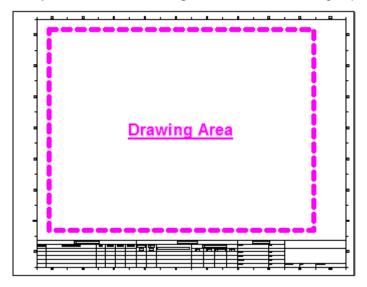
Place Drawing Area Command

Places a drawing area for drawing border templates. This command is available when you are editing a drawing border template for which a drawing area is not defined. If a drawing area already exists, an error messages displays. You can delete the existing drawing area and place a new one.

A drawing area is a property of the border template that defines the "useful" area of a drawing sheet. During the creation of a new composed drawing, the layout template containing the managed views fits inside the drawing area of the border template.

By defining a drawing area in each border template, a single layout template can be fit into any size border. The proportion of the drawing area devoted to a particular region in the layout does not change when the border template changes. For example, if you apply a region to the bottom half of a layout (and the layout is merged into a border template), the region always consumes the bottom half of the drawing, regardless of how it is stretched or resized by different border templates.

After you click Place Drawing Area III, click and drag to place the drawing area.



Save the border template file and exit SmartSketch Drawing Editor. You can use the border file to create new 3D composed drawings. For more information, see the *Drawings and Reports User's Guide* or the *Common User's Guide*.

■ NOTES

- Editing a border template on the Symbols share does not affect an existing drawing because the border template associated to a drawing is stored in the database. To change the border template used by an existing drawing, you must replace the stored border template using the Switch Border command.
- If no drawing area exists in the border template used to create a composed drawing, one is computed based on the white space in the border template. This computed drawing area is not saved with the border template.
- The existence of a drawing area in the border template associated to a drawing has an impact on the behavior of the Switch Border command.

See Also

Switch Border (on page 210)

Change the Border for an Individual Drawings by Rule Sheet

The following procedure steps you through changing the border for an individual sheet to a template different from the border template defined in the Drawings by Rule component. For example, you may want the first sheet of drawing to use a different border template than all other sheets.

■ NOTE For more information on **SmartSketch Drawing Editor** commands used in this procedure, see the *SmartSketch Drawing Editor Help*.

Change the Border Template for an Existing Sheet

- 1. Right-click a drawings by rule document in the Detail View and select **Edit**.
 - The drawing opens in SmartSketch Drawing Editor.
- 2. Select a sheet tab at the bottom of the drawing view.
- 3. Click **Edit Sheet Properties** A on the toolbar.
 - The **Sheet Properties** dialog box displays.
- 4. On the **Current Sheet** tab, select a new value for **Border Template**.
- 5. Click OK.

The new border template is applied to the sheet.

NOTE Layout Template is inactive, and cannot be changed for an existing sheet.

Apply a Border and Layout Template to a New Sheet

- 1. Right-click a drawings by rule document in the Detail View and select **Edit**.
 - The drawing opens in **SmartSketch Drawing Editor**.
- 2. Right-click a sheet tab at the bottom of the drawing view and select Insert.
 - A new sheet is created and the **Sheet Properties** dialog box displays.
- 3. On the Current Sheet tab, select a value for Border Template and Layout Template.

4. Click OK.

The border and layout templates are applied to the new sheet.

NOTE A sheet cannot be deleted unless all views on the sheet are removed first.

Edit Layout Template

Available when you click **Tools > Edit Layout Template** in the Drawings and Reports task.

Edit Border Family Command

Allows you to change the border family associated with the current layout template. This command is available when you edit a layout template.

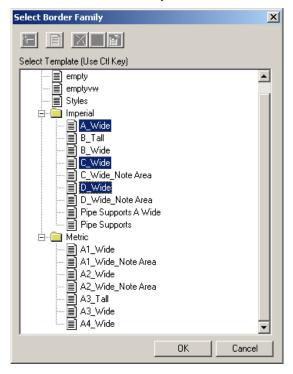
When you click **Edit Border Family** A, the **Select Border Family** dialog box appears so that you can select border templates to associate to the current layout template.

See Also

Select Border Family Dialog Box (on page 398)

Select Border Family Dialog Box

Specifies the border templates to associate with the current layout template. This dialog box displays when you click **Edit Border Family**, which is only available in **SmartSketch Drawing Editor**. You can multi-select borders by holding down the CTRL key while selecting borders from the hierarchy.

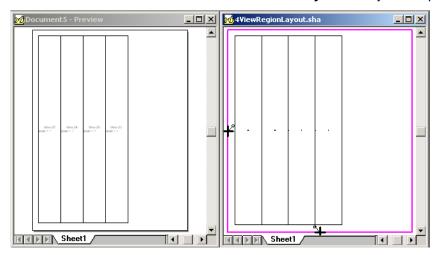


See Also

Edit Border Family Command (on page 398)

Preview Layout Command

Shows a preview of the drawing document based on the current layout. A preview is shown for each of the borders chosen in the **Border Family** for the layout template.



This command is only available in **SmartSketch Drawing Editor** when you edit a layout template.

Shortcut Menus

Commands available when using right-click shortcut menus.

Update View Command

Updates the contents for the selected drawing, report, key plan, snapshot, or detail/section view. This command is available for composed drawing views only. Right-click a selected view and select **Update View** on the shortcut menu. The view contents update against the current 3D model. When working with the **Update View** command:

- If the software encounters a problem before or during the drawing update, it stops updating, displays either an error status or error message, and saves the errors to the log file. For more information, see Conditional Drawing Update (on page 86) in the Orthographic Drawings User's Guide.
- You cannot multi-select views. The command is not available on the shortcut menu if more than one view is selected.
- The inputs for the view (the associated volume or drawing view) must be defined and valid.
- If a report is defined as part of a drawing view definition, the **Update View** command is not available.
- If the drawing document is read-only, the Update View command is not available.

■ NOTES

■ To remove associated inputs from a view, use the **Remove Associated Inputs** command. For more information, see *Remove Associated Inputs Command* (on page 319). To associate a view to a volume or another drawing view, use the **Associate Objects to**

View command. For more information, see Associate Objects to View Command (on page 300).

• For composed drawings, views that are too small to display the volume are automatically sized larger on update to fit unless the view is set to **Fit to Scale** or is managed by a region. Likewise, views that are larger than the volume are automatically resized to fit unless the view is set to **Fit to Scale** or is managed by a region. The view size grows or shrinks from the center of the view and view proportions may change after the resize. The drawing must be saved to make the resize of the view permanent. For more information, see *Automatic Resize Behavior of Composed Views* (on page 285).

See Also

Place View Command (on page 275)
Associate Objects to Views (on page 314)
Place Snapshot View Command (on page 299)
Place Report View Command (on page 319)

Switch Border

Allows you to switch the border template associated with the composed drawing documents selected in the Detail View. This command is available on the shortcut menu when you select a composed drawing document. It displays the **Drawing Sheet Properties** dialog box so you can change the associated border template.

Editing a border template in the SharedContent folder does not affect an existing drawing because the border template associated to a drawing is stored in the database. To change the border template used by an existing drawing, you must replace the stored border template using the **Switch Border** command.

After switching the border template, the software computes a new position and size for any regions and managed views contained in the drawing. The shift and resize of the view is proportional to the size of the drawing areas in the border templates. If the border template does not contain a drawing area, the drawing boundary is computed automatically.

The software does not distinguish between different types of drawing views (report, key plan, and graphic views) when switching the border.

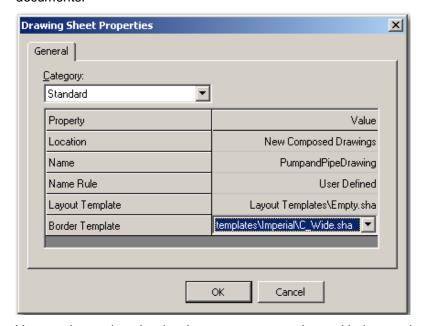
■ NOTES

- If you switch the border template of a drawing, views may resize or reposition with the new border template based on the following conditions:
 - Managed views are proportionately resized and repositioned according to the size of the new border template.
 - Unmanaged views are not resized and repositioned.
- Drawings must be updated after switching the border template in order to ensure all automated annotation is positioned correctly on the drawing.
- For views that are resized after the border template switch, drawing views using Fit to Scale show the same content. Views with a scale may have content clipped out if the view is made smaller after the switch.

For more information, see *Place Region Command* (on page 296).

Drawing Sheet Properties Dialog Box

Shows the drawing sheet properties associated with the selected drawing document(s) and allows you to change the border template file. When displayed for the **Switch Border** command on selected composed drawing documents, the only property available is **Border Template**. Select **More** in the **Value** field to display the **Select Template** dialog box and specify a new border template. Click **OK** to associate the new border template file to the selected drawing documents.



You need to update the drawings to regenerate them with the new border template file.

See Also

Switch Border (on page 210)

Add a Sheet to Drawing

In this workflow, you change the border for an individual sheet to a template different from the border template defined in the component. For example, you may want the first sheet of drawing to use a different border template than all other sheets.

- Right-click a drawings by rule document in the Detail View and select Edit.
 - The drawing opens in **SmartSketch Drawing Editor**.
- 2. Right-click a sheet tab at the bottom of the drawing view and select Insert.

A new sheet is created and the **Sheet Properties** dialog box displays.



- 3. On the **Current Sheet** tab, select values for **Border Template** and **Layout Template**. Click **Use Default** to apply the default values for the drawing, as shown on the **Default** tab.
- 4. Click OK.

The border and layout templates are applied to the new sheet.

Layers (SmartSketch Drawing Editor Tools Menu)

Layers are used to distinguish between graphics within a template or drawing. You can create layers in **SmartSketch Drawing Editor** with the **Tools > Layers** command.

Objects like drawing views, key plan views, report views, and drawing property labels are placed on the **DwgTemplate** layer when you save the drawing document. You should not place manual markups on the **DwgTemplate** layer.

When placing manual markups, such as graphics or company logos, place them in paper space within the drawing layers, not in *model space* (inside a drawing view). Markups placed within *model space* are not preserved. When a graphic rule does not specify a layer, intelligent graphics are placed on the **Default** layer.

Composed Drawings

When you create composed drawings, you can place manual markups on any layer of the drawing.

Volume (Spatial) and Orthographic Drawings by Query Drawings

For volume drawings, manual markups are preserved on any layer of the drawing, except the **DwgTemplate** layer. The **DwgTemplate** layer is reserved for system use. If you placed manual markups on the **DwgTemplate** layer, they will be lost when you update the drawing document.

Isogen Isometric Drawings

If you intend to create manual markups within an Isogen Isometric drawing, place your markups on the **Default** layer. The software preserves this layers when you update drawings. Other layers are not preserved.

If named layers do not exist in the template, the software creates them using the symbology specified in the style XML file. In ISOGEN Configuration, use the options in the **Drawing Area > Graphics > Layers** style group to create new layers within the style XML file. Map definitions to the layers under **Drawing Area > Graphics > Definitions**.

If the named layers do exist in the template, use **Tools > Display Manager** in **SmartSketch Drawing Editor** to change the symbology used within the template.

Modify an Existing Border File

You can create an isometric drawing border from scratch using **SmartSketch Drawing Editor**. You use the commands available within **SmartSketch Drawing Editor** to place graphics and create appropriate layers.

You should name new border files with the name of the needed isometric style, such as Iso_Pipeline or Iso_Piperun.

1. Navigate to [Reference Data Folder]\SharedContent\PmfglsoStyleData folder, and copy the appropriate existing border igr file to the [Reference Data

Folder]\SharedContent\Drawings\Catalog\Templates folder. You can create a subfolder for the new file.

- 2. Rename the copied file with the .sha extension.
- 3. Open the copied .sha files with SmartSketch Drawing Editor.
- 4. Fit the view.
- 5. Select Tools > Layers.

The Layer ribbon displays.

- Select Tools > Display Manager. On the Layers tab, scroll down to see the values
 currently set in the .sha file. These are the color, line type, and width values for the named
 layers.
- 7. Make changes as needed, save the file, and exit SmartSketch Drawing Editor. The next time you use this .sha file as your drawing border template, the graphics will show the changes you made.

Custom Commands

Custom commands used in SmartSketch Drawing Editor.

Convert Excel Spreadsheet Reports to Native Text Box Format Custom Command

The **SP3DConvertExcelEmbedded.dll** is a delivered custom command that allows you to convert an Excel spreadsheet report to the native text box format for use in 3D Drawings. The DLL to execute this command is located on the client machine in the [Product Folder]\Drawings\Client\Bin folder. To run this custom command, select **Tools > Custom Commands** in an open drawing in SmartSketch Drawing Editor, and browse to the DLL location on the computer. For additional information on any of the **SmartSketch Drawing Editor** commands mentioned in this procedure, see the *SmartSketch Drawing Editor Help*.

To convert an embedded report, you must set the properties correctly on the associated views. For a 3D composed drawing, set the **Report Output** property for a report view as needed. For more information, see *Report Properties Dialog Box (Place Report View Command)* (on page 320). For a 3D volume drawing, select the **Convert report output to text boxes (no Excel)** property for the drawing view associated to the report view. For more information, see *View Tab (Drawing View Properties Dialog Box)* (on page 383).

When you run this command, the **Convert** dialog appears so you can specify the appropriate options for the convert operation:

Convert embedded Excel spreadsheet

Indicates that the specified report will be converted to native text box format.

Convert and embed an Excel spreadsheet

Indicates that the specified report will be converted to native text box format and then embedded in the current drawing document.

Replace converted spreadsheet with a new converted version from Excel

Specifies that a converted spreadsheet report will be replaced with a new version of the same report.

File

Indicates the report file to convert. Click the ellipsis button to browse to the appropriate file.

See Also

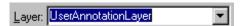
Region Properties Dialog Box (on page 298)

Edit a Composed Drawing

This procedure steps you through editing a composed drawing from either the Drawings and Reports task **Management Console** or from a 3D modeling task using the **Tools > Drawings Console** command. For additional information on any of the **SmartSketch Drawing Editor** commands mentioned in this procedure, see the *SmartSketch Drawing Editor Help*.

- Right-click an existing composed drawing in the Management Console or the Drawings Console. The document opens in SmartSketch Drawing Editor. You can use the editing tools in SmartSketch Drawing Editor to modify the appearance of your drawing. You can also use the Composed Drawing tools to place new regions, drawing views, report views, and snapshot views on the drawing area. You can also place labels manually on the drawing.
 - NOTE When you place objects like drawing views, key plan views, report views, and drawing property labels, the software automatically places them on the **DwgTemplate** layer when you save the drawing document. You should not place manual markups on the **DwgTemplate** layer.

If you use other native **SmartSketch Drawing Editor**. commands (such as **Place Line** or **Place Dimension**) to add manual markups to the template, put them on the **Default** or a layer with "User" as the prefix (for example, a layer named **UserAnnotationLayer** to preserve the changes when you update drawings. For more information on layers, see *Layers* (*SmartSketch Drawing Editor Tools Menu*) (on page 402).



- 2. To place a new region in the drawing area, use the **Place Region** command. Regions are used to manage drawing views. Click and drag to place the new region. The **Region Properties** dialog box displays after you place the region shape. Specify the properties for the region you are adding and click **OK** to complete the creation.
- 3. To place a new drawing view, use the Place View command. If you want your new region to manage this drawing view, place the new drawing views within or touching the region. You can also place "unmanaged" views outside existing regions. Click and drag to create the drawing view shape. The Drawing View Properties dialog box displays so you can define the drawing view. For more information on the Drawing View Properties dialog box, see Drawing View Properties Dialog Box (Place View Command) Steel Order Drawings (on page 281). For more information on how regions and views work together, see Composed Drawings (on page 201) in the Orthographic Drawings User's Guide.

4. To place a report view, click **Place Report View** . The drawing view you select can be managed by a region or unmanaged (outside all regions). Click and drag to place the report on the drawing area. The **Report Properties** dialog box displays so you can specify the properties for the report. Click **More** in the **Report Template** property box to select from available reports. You can set the output format of the report using the **Report Output Format** and **Report Justification** properties on this dialog box. For more information, see *Report Properties Dialog Box (Place Report View Command)* (on page 320).

■ NOTES

- The report view you create is empty. Use the **Associate Objects to View** command to associate a drawing view to the new report view. For more information, see *Associate Objects to View Command* (on page 300). The drawing view you select must already be associated with a volume in the model.
- You can associate multiple reports to the same view, but you cannot, in the current release, associate multiple views to the same report.
- 5. You can also place snapshot views on your composed drawing. You create the snapshot view content in a 3D task using the **Tools > Snapshot View** command. For more information on creating the snapshot view, see **Snapshot View Command** in the *Common User's Guide* available from **Help > Printable Guides**.
 - To place the snapshot view on the composed drawing, use the **Place Snapshot View** command. Click and drag to place the snapshot view shape. The **Drawing View Properties** dialog box displays so you can specify the properties for the snapshot view.
- 6. You can place manual labels on the composed drawing using the **Place Label** command. For more information on placing labels manually, see *Place a Manual Label* (on page 325).
- 7. If a view association is incorrect, click **Remove Associated Inputs** . This command allows you to select a view and remove all associations to volumes, folders, and other views
- 8. If you decided to delete a drawing view, a message box displays when you press **Delete** for a selected view, providing a chance to cancel the operation. After a drawing view is deleted from the drawing, the operation cannot be reversed.
- 9. To update the contents of a view, select and right-click the view then select **Update View** on the shortcut menu. The view is updated with the associated 3D model objects or report, depending on the association you made with the **Associate Object to View** command.
- 10. Save your drawing changes before exiting **SmartSketch Drawing Editor**.
- 11. Update the modified drawing to incorporate the changes. You can open the drawing to check the new layout and view content.

See Also

Composed Drawings Common Tasks (on page 203) Create a new composed drawing (on page 205)

Edit the Drawing Template

- 1. Right-click the component and select **Edit Template**. SmartSketch Drawing Editor and the **Sheet Properties** dialog box displays.
- On the Sheet tab, select values for Sheet Assignment Rule, Sheet Naming Rule, Border Template, and Layout Template. These values are used for the first sheet when a drawing document is created.
- 3. On the **Document** tab, select values for **Document Assignment Rule**, **Document Naming Rule**, **Border Template**, and **Layout Template**. These values are used for sheets added to a drawing document after it is created.
 - ▶ NOTE You can also click Edit Sheet Properties on the Edit Template ribbon toolbar to change values in the Sheet Properties dialog box.
- 4. Click File > Save.
- 5. Click File > Exit. The SmartSketch Drawing Editor window closes.

Modify View Ribbon

Modifies the view size. This ribbon displays after a view is selected in SmartSketch Drawing Editor.

Finish

Saves changes to the view.

Update View

Select to update the view (Drawing) gets updated along with the parent volume in the model.

Undo Crop

Select to remove an existing crop on the view. The view returns to its original boundaries. This option is not available for a view that cannot be cropped, such as a composed drawing view.

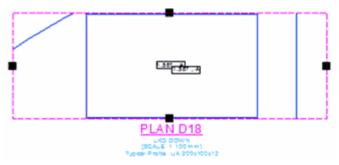
▶ NOTE Removal of cropping with **Undo Crop** is not visible on a view until the view is updated. Select **Update View**, and click **Finish** to see the results immediately. The results will also be visible after using **Update** or **Batch Update** in the Drawings and Reports task at a later time.

Crop a Drawings by Rule 2D Drawing View and the 3D Model Volume

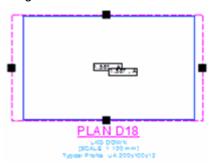
When an orthographic steel order view in a drawings-by-rule drawing is based on a volume in the model, you can crop the view, and also resize the model volume to match the cropped view.

1. Select the view in SmartSketch Drawing Editor.

Four handles display on the view frame and **Modify View** ribbon displays.



2. Drag the handles to resize the view to the needed size.



- 3. To immediately update the view, select **Update View**.
- 4. Click Finish.

The cropped view is saved and parent volume in the model is resized.

TIPS

- To cancel the changes to the view size, click outside of the view without clicking Finish.
- If **Update View** is not selected, the view is marked out-of-date in the **Drawings View Explorer**. You can right-click the view and select **Update**.

■ NOTES

- The 2D view and the 3D volume maintain the cropped size with subsequent updates.
- Shell expansions are not based on a volume and cannot be cropped. For more information, see Create a shell expansion drawing (on page 173) in the Orthographic Drawings User's Guide.
- Section and Detail views use their own commands for cropping. For more information, see Place a Section View (on page 355) and Place a Detail View (on page 359).

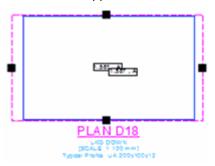
See Also

Drawings View Explorer (on page 256)

Remove Cropping on a Drawings by Rule View

1. Select a previously-cropped view in **SmartSketch Drawing Editor**.

Four handles appear on the view frame and **Modify View** ribbon displays.



- 2. Select Undo Crop and Update View.
- 3. Click Finish.

The view (and the parent volume in the model) changes to its original size.



Place an Unassigned View

The following steps describe the basic workflow for placing an existing view that is not assigned to a drawing.

- 1. Right-click a drawing document in the Detail View and select **Edit**. The drawing opens in **SmartSketch Drawing Editor**.
- 2. Drag an unassigned view if from the **Unassigned Folder** in the Drawings View Explorer to the graphics view of the drawing.

The view displays in the graphics view and the view icon (or or or displays under the drawing sheet in the Drawings View Explorer.

■ NOTES

Dragging more than one view onto an existing drawing may require:

- Deleting the existing drawing region, and dragging the views onto the drawing without a region.
- Deleting the existing drawing region, and adding a new region that supports the number of views needed. For more information, see Edit Template (Drawings by Rule) (on page 128) in the Orthographic Drawings User's Guide and Layout Style Rules in Orthographic Drawings in the Drawings and Reports Reference Data Guide.

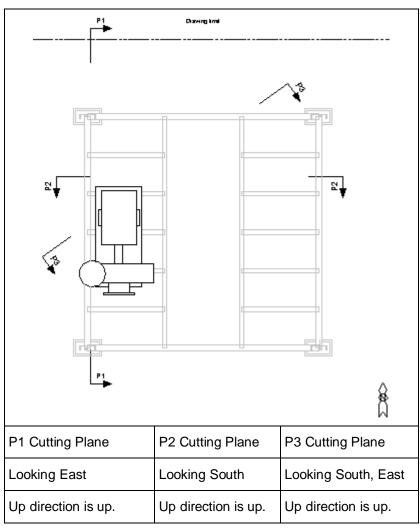
Section View Orientation Rules in Drawings by Rule

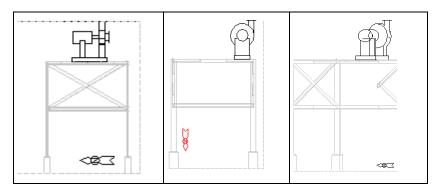
Plant Mode Section View Orientation Behavior

The software automatically determines the "up" direction for section views. In most cases, the up direction is "up," or +Z. In cases where using +Z is impossible (for example, if the section view direction is Looking Plan or Looking Up), the software orients the view to the North direction.

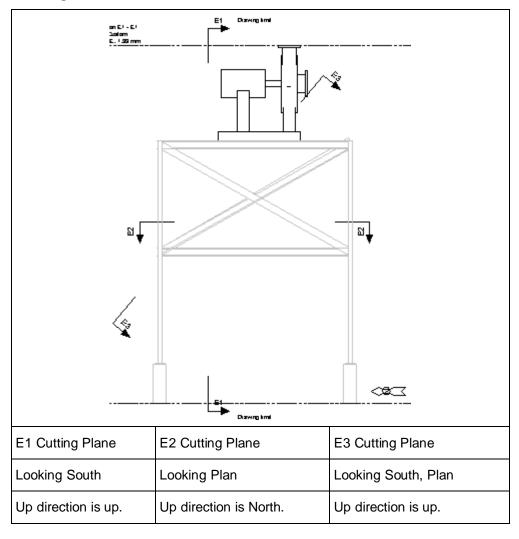
The following examples outline common section view orientations:

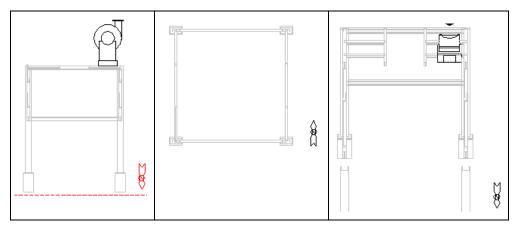
Looking Plan



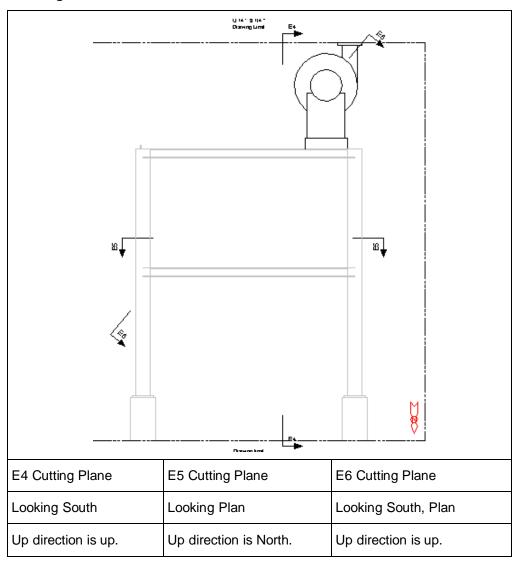


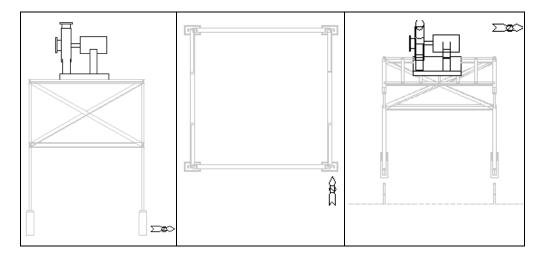
Looking North





Looking East





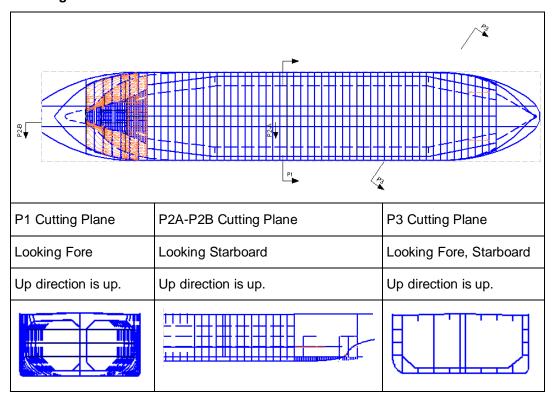
Marine Mode Section View Orientation Behavior

The software automatically determines the "up" direction for section views. In most cases, the up direction is "up," or +Z. In cases where using the z-axis is impossible (for example, if the section view direction is Looking Plan or Looking Up), the software orients the view to the Port direction.

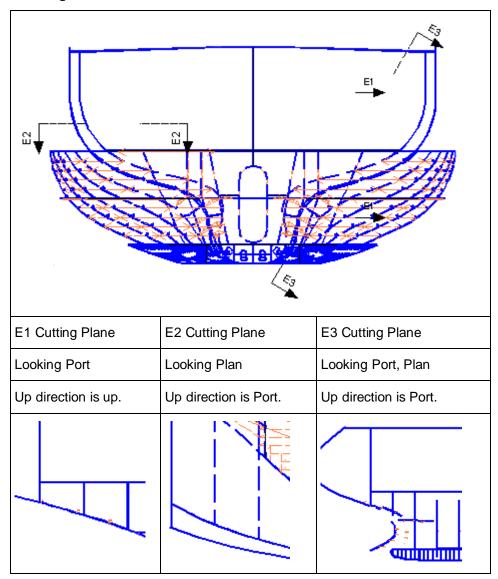
Additionally, angled sections of elevation views, and any section of a non-standard view direction (in which it is possible) uses Port orientation.

The following examples outline common section view orientations:

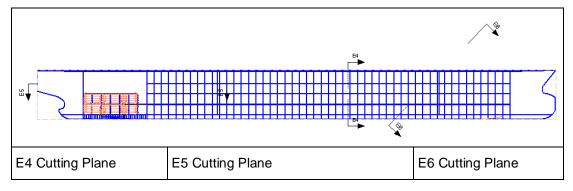
Looking Plan

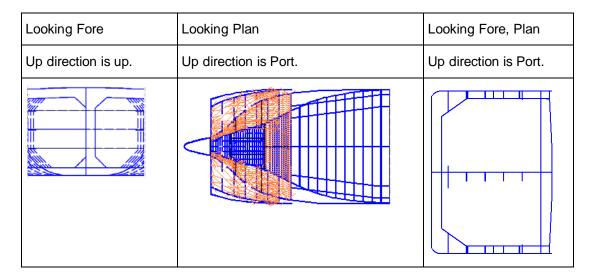


Looking Aft



Looking Port





For more information, see *Orientation Rules in Drawings by Rule* in the *Drawings and Reports Reference Data Guide*.

Update and Full Update Commands

The **Update** and **Full Update** commands are available when you right-click a drawing view in the **Drawings View Explorer**.

Full Update performs a full update of all geometry in the view. It does not consider whether the model objects associated with the geometry has changed.

Update performs a smart update of the view or a set of selected views from the same sheet. If a set of criteria is met, then an incremental update is performed only for the geometry of added, modified, and deleted objects. If the criteria are not met, then a full update — the equivalent of **Full Update** — is performed.

All of the following criteria must be met for **Update** to trigger an incremental update:

- The view has had at least one full update previously performed.
- The only view properties that have changed are name and description. For more information, see View Tab (Drawing View Properties Dialog Box) (on page 277).
- The numbers of added, modified, or deleted objects are less than set quantities.
- The software allows incremental updates for the selected view type.

Any one of the following criteria cause **Update** to trigger a full update:

- The view is new and has never had a full update.
- The view is a section or detail view. For more information, see *Place Detail Envelope Command* (on page 343) and *Place Cutting Plane/Section View Command* (on page 346).
- The view has been cropped. For more information, see Crop a Drawings by Rule 2D Drawing View and the 3D Model Volume (on page 406).
- The view style, scale, orientation, or coordinate system have changed. For more information, see View Tab (Drawing View Properties Dialog Box) (on page 277).

- A primary plate object in the view has been spit or unsplit.
- The numbers of added, modified, or deleted objects are greater than a set percentage of the total number of objects.
- The software does not allow incremental updates for the selected view type.

■ NOTES

- An incremental update is usually faster than a full update, but still results in completely up-to-date geometry for the view.
- The View Log command displays a log of the results from the last update performed on the view with the Update or Full Update commands.

See Also

Drawings View Explorer (on page 256)

Run Update or Full Update

You run **Update** or **Full Update** after structure displayed in a view has been modified in other tasks, such as Molded Forms or Structural Detailing.

1. Right-click a drawing document in the **Drawing Console** or the Detail View of the Drawings and Reports task, and select **Edit**.

The drawing opens in **SmartSketch Drawing Editor**.

2. If you know that objects in a view have changed, even if the view is still marked as up-to-date in the **Drawings View Explorer**, right-click the view in the **Drawings View Explorer** and select **Update** or **Full Update**.

A progress bar displays, detailing the update process.

NOTE You can update multiple views in the same sheet by selecting the sheet node or the individual sheets.

- 1. If you do not know if objects in a view have changed:
 - Right-click the view in the drawing area or the **Drawings View Explorer**, and select
 Refresh. The view is marked *out of date* in the **Drawings View Explorer** if objects have changed.
 - Right-click the view in the Drawings View Explorer, and select Update or Full Update.
- 2. Optionally, right-click the view in the **Drawings View Explorer**, and select **View Log** to check the results of the update.

Delete Views

Delete Views Placed by the Place View Command

- 1. Right-click a view placed by using Place View X.
- 2. Select Delete.

The **Delete Views** dialog box displays.

3. Select **Yes** to delete the view permanently. To cancel the operation, click **No**.

Delete Views Placed by Rules

- 1. Right-click a view in Drawing View Explorer.
- 2. Select Delete.

The Delete Views dialog box displays.

3. Select **Delete** to delete the view permanently.

■ NOTES

- The UnAssign option moves the view to the UnAssigned Folder. The Cancel option cancels the operation.
- If you delete a view that is a parent of other views, such as a detail or section view, the Convert or Delete dialog box displays. Select Convert to independent drawing view(s) to save the child view as an independent drawing view, or select Delete to delete the child view.

Delete Views from the UnAssigned Folder

- 1. Right-click a view from the **UnAssigned Folder**.
- 2. Select Delete.

The **Delete Views** dialog box displays.

3. Select **Yes** to delete the view permanently. To cancel the operation, click **No**.

SECTION 20

Save as SmartPlant Review File

You can view your 3D Model Data in SmartPlant Review (SPR) even if your model has not been registered using the SmartPlant Registration Wizard. When you create a 3D Model Data component, you set it up to save as a SmartPlant Review file. This enables the **Save As SmartPlant Review** command on the right-click menu for the 3D Model Data documents after they are created and up-to-date.

The **Save As SmartPlant Review** command is not available if you setup the 3D Model Data component to save to **Disk Only** or if you selected the **Generate CAD output (.sat file)** option on the **Setup** dialog box.

Before you can use the **Save As SmartPlant Review** command, you must install the SmartPlant Schema Component on the local client machine. For more information, see the *Intergraph Smart*TM 3D *Installation Guide*, available from the **Help > Printable Guides** command.

SPR shows the objects from the VUE file using global coordinates. Before you save the 3D Model Data component documents as SmartPlant Review (SPR) files, right-click the component and select **Properties** to check the **Style** tab **Coordinate System** property setting. You want to make sure the **Plant Monument Coordinate Offset** is passed correctly to SPR when creating the VUE file. The offset value allows you to see the original coordinates relative to the new SPR coordinate system.

The 3D Model Data documents use surface styles and aspects as part of their property definition so that they can be used in SPR. You set the properties at the 3D Model Data document level. For more information, see *Set surface styles and aspects for 3D model data documents* (on page 48).

The **Save as SmartPlant Review** command saves a 3D Model Data document as a SmartPlant Review file. The file can then be viewed in SmartPlant Review. This command is available when you right-click an up-to-date 3D Model Data document if it was setup to save as a SmartPlant Review file. For more information, see *Setup a 3D Model Data component* (on page 191). The **Save As SmartPlant Review** command is not available if you set up the 3D Model Data component to save to **Disk Only** or if you selected the **Generate CAD output (.sat file)** option on the **Setup** dialog box.

The command creates both a property data file in XML format and a graphics file in VUE format (.vue).

For more information, see Save as SmartPlant Review Dialog Box (on page 419).

Recommendations for Exporting to SmartPlant Review

- The number of objects generated by a 3D Model Data component and exported successfully to a SmartPlant Review file depends largely on the type of objects and your hardware resources. The more objects in your file, the more processing time the software requires. It is possible to have so many objects in the file that the software uses up all of the available memory. Larger models can take hours to process, so batch updating is recommended.
- Monitor the error logs regularly for resource issues, even if the specified filter worked initially. You can add more objects to the model meeting the filter criteria.

- SmartPlant Review (SPR) version 6.1.0.15 (or higher) allows you to open multiple VUE files simultaneously. Refer to your SmartPlant Review documentation for more information. When you open VUE and XML files in SPR for the first time, SPR builds a database containing the tag information for the files. This process may take a significant amount of time. After the multiple VUE files are opened in SPR, you can create SVF files for future loading of VUE files.
- SmartPlant Review (SPR) version 6.2.0.29 (or higher) supports turning Smart 3D aspects on and off. All aspects are turned on by default in SPR. The SPRSchema.txt file can be customized to add aspects.

■ NOTES

- You can use SmartPlant Review to review each resulting model of Save as SmartPlant Review action. However, you cannot review multiple models.
- Before you can use the Save as SmartPlant Review command, you must install SmartPlant Schema Component on the local client machine. For more information, see the Intergraph SmartTM 3D Installation Guide, available from the Help > Printable Guides command.
- The Save as SmartPlant Review command also looks for the EFSchema file (P3DComponent.xml). If your model has been registered using the SmartPlant Registration Wizard, the software retrieves the file from the integrated environment. If your model has not been registered using the SmartPlant Registration Wizard, the software looks for the file in the SharedContent share.
- You can publish .zvf files and launch SmartPlant Review to view the .zvf files. You can also view the .zvf file using View and Markup if you have SmartPlant Markup Plus installed. You do not have to use the Save as SmartPlant Review command to use the SmartPlant Review features.
- The 3D View Control used for viewing the published graphics in SmartPlant Markup Plus does not currently support turning aspects on and off.

Save 3D model data for SmartPlant Review

Before you can use the **Save as SmartPlant Review** command, you must install the SmartPlant Schema Component on the local client machine. For more information, see the *Intergraph Smart*TM *3D Installation Guide*, available from the **Help > Printable Guides** command. The **Save as SmartPlant Review** command also looks for the EFSchema file (*P3DComponent.xml*). If your model has been registered using the SmartPlant Registration Wizard, the software retrieves the file from the integrated environment. If your model has not been registered using the SmartPlant Registration Wizard, the software looks for the file in the SharedContent share.

The **Save as SmartPlant Review** command is available on the right-click menu for the 3D Model Data documents when they are set up to save as a SmartPlant Review file. For more information, see *Setup a 3D Model Data component* (on page 191).

Recommendations for Exporting to SmartPlant Review

The number of objects generated by a 3D Model Data component and exported successfully to a SmartPlant Review file depends largely on the type of objects and your hardware resources. We recommend you limit each 3D Model Data component filter to approximately 30,000 objects if using Smart 3D version 6.1 Service Pack 1 or later (approximately 20,000 objects is using an earlier version).

- Monitor the error logs regularly for resource issues, even if the specified filter worked initially. You can add more objects to the model meeting the filter criteria.
- SmartPlant Review (SPR) version 6.1.0.15 (or higher) allows you to open multiple VUE files simultaneously. Refer to your SmartPlant Review documentation for more information. When you open VUE and XML files in SPR for the first time, SPR builds a database containing the tag information for the files. This process may take a significant amount of time. After the multiple VUE files are opened in SPR, you can create SVF files for future loading of VUE files.
- SmartPlant Review (SPR) version 6.2.0.29 (or higher) supports turning Smart 3D aspects on and off. All aspects are turned on by default in SPR. The SPRSchema.txt file can be customized to add aspects.

■ NOTE The 3D View Control used for viewing the published graphics in SmartPlant Markup Plus does not currently support turning aspects on and off.

- 1. Before you save your 3D Model Data component documents as SmartPlant Review files, right-click the component and select **Properties** and go to the **Style** tab to make sure the **Coordinate System** property is set appropriately so that the **Plant Monument Coordinate Offset** is passed correctly to SPR when creating the VUE file. This is because SPR shows the objects from the VUE file using global coordinates. The offset value allows you to see the original coordinates relative to the new SPR coordinate system. For more information on 3D Model Data components, see 3D Model Data (on page 188). For information on saving to SPR, see Save as SmartPlant Review File (on page 417).
- You should also make sure that the Surface Styles and Aspects properties are set correctly on the 3D Model Data documents before saving them for viewing in SmartPlant Review.
 - Set surface styles and aspects for 3D model data documents (on page 48)
- Right-click an up-to-date 3D Model Data document and select Save As SmartPlant Review.
- 4. On the **Save as SmartPlant Review** dialog box, specify a property data file. This file will be an XML format file.
- 5. Specify a graphics file. This file will be a VUE format file (.vue).
- 6. Click **OK** save the SmartPlant Review files to the specified names and locations.

Save as SmartPlant Review Dialog Box

Specifies the files needed to save documents for viewing in SmartPlant Review.

NOTE You can publish .zvf files and launch SmartPlant Review to view the .zvf files. You can also view the .zvf file using **View and Markup** if you have SmartPlant Markup Plus installed. You do not have to use the **Save as SmartPlant Review** command to use the SmartPlant Review features.

Data file

Specifies the XML file that contains all the objects and property data that the application publishes for viewing and that meet the filter query specified for the 3D Model Data component. For more information on the property data that is published, see *The Integration Reference Guide* located in **Help > Printable Guides**.

Graphics file

Specifies the VUE file (.vue) to which the document graphics are saved. This is the file you would select in SmartPlant Review with the **File > Open** command.

You can browse for either file using the ellipsis button next to the field.

See Also

Save as SmartPlant Review File (on page 417) Save 3D model data for SmartPlant Review (on page 418)

SECTION 21

Revising

The document revision process is separate from the publishing process, making it possible to revise a document locally and save it to the database without re-publishing it. The **Revise** command is available on the right-click menu for drawings, reports, and 3D Model Data documents. In an integrated environment, all revisions are handled by SmartPlant Foundation.

Revising and publishing are two separate actions. You specify the document revision using the **Revise** command, which creates a Revision for the document with Major and Minor set, depending on the revision schema selected. If you are working in an integrated environment, you can modify the other revision information on the document.

After setting the revision number, right-click the document and select **Properties**. Select the **Revision** tab and edit the **Revision** fields. You should update documents to include any new title block information.

You can now re-publish the document with the new revision information.

■ NOTES

- You can use the Revise command if your model has been registered using the SmartPlant Registration Wizard. For more information on registering, see the *Project Management* User's Guide under Help > Printable Guides.
- If the drawing document that you are looking at in the **Detail View** has a yellow icon (for example: (i), the drawing document is a version 6.1 legacy Snapshot drawing. You should use the **Tools > Convert Legacy Snapshots** command to convert this document to a Composed Drawing for use in the current version of the software. If you do not convert the legacy snapshot drawing, you cannot perform edit operations on the drawing, including update, revise, and publish.

See Also

Publishing Documents (on page 424) Revise (on page 421) Revise a document (on page 422) Convert Legacy Snapshots (on page 240)

Revise

Revising and publishing are two separate actions. You specify the document revision using the **Revise** command, which creates a Revision for the document with Major and Minor set, depending on the revision schema selected. If you are working in an integrated environment, you cannot modify the Major and Minor revision data, but you can modify the other revision information on the document.

After reserving the revision number, right-click the document and select **Properties**. Select the **Revision** tab and edit the **Revision** fields. You should update documents to include any new title block information.

You can now re-publish the document with the new revision information.

Revise Dialog Box (on page 422)

■ NOTE You can use the **Revise** command if you have registered your model using the SmartPlant Registration Wizard. For more information on registering, see the *Project Management User's Guide* under **Help > Printable Guides**.

Revise a document

You can revise drawings, reports, and 3D Model Data documents if you have registered your model using the SmartPlant Registration Wizard. For more information about using the SmartPlant Registration Wizard, see the *Project Management User's Guide* under **Help > Printable Guides**.

1. Right-click a document and select **Revise**. The **Revise** dialog box displays.

TIPS

- You can also multi-select documents in the **Detail View**, or you can select a folder in the **Management Console** hierarchy to select all of the documents within the folder if they all have the same revision level.
- If the Revise command is not available on the shortcut menu, check the properties on the document. Right-click the document and select Properties. Go to the WBS tab and make sure that you have a Document type and Discipline set for the document. For more information, see Set properties for publishing documents (on page 428).
- 2. For a new document or a document that does not yet have a defined revision scheme, select the revision scheme that you want to use from the **Revision Scheme** list.
 - **NOTE** Only revision schemes that are applicable to the configuration (plant) or classification (document type) are available in the shortcut menu. The revision schemes related to a configuration or classification are not available for any other configurations or classifications. If none of the revision schemes are related to the configuration or classification, then all revision schemes are available unless they are related to any other configuration or classification. For more information on revision scheme configuration, see *Configuring Different Revision Scheme Strategies* in the *How to Configure Document Management* guide.
- 3. In the **Revise in Tool** section, select the next available major and minor revision numbers.
- Click **OK**. The document is saved to the model database. The command creates a revision record by adding it to the document Revision properties. The command also reserves the revision number.
- 5. Right-click the document and select **Properties**.
- 6. Go to the **Revision** tab and edit the values in the new revision row.
- 7. Update the document to update any document property title block information. For more information, see *Updating Documents* (on page 78).
- 8. Re-publish the document. The stored document is not updated until you publish it.

Revise Dialog Box

Allows you to revise a document in the database of the authoring tool without publishing it.

NOTE Fields with a shaded background are read-only fields and cannot be edited.

Selected documents

Displays a list of the documents selected to be revised or for which you want to reserve a

set of revision numbers. You populate this list by selecting documents before you use the **Revise** command.

Engineering Tool

Opens an authoring tool-specific dialog box that allows you to select documents to add to the **Selected documents** list. This option is not available in Smart 3D.

Revision Scheme

If you have selected a new document or a document for which no revision scheme has been selected, choose the revision scheme to be applied from the list of available options. Only revision schemes that are applicable to the configuration (plant) or classification (document type) are available in the shortcut menu. The revision schemes related to a configuration or classification are not available for any other configurations or classifications. If none of the revision schemes are related to the configuration or classification, then all revision schemes are available unless they are related to any other configuration or classification. For more information on revision scheme configuration, see *Configuring Different Revision Scheme Strategies* in the *How to Configure Document Management* guide.

Current Revision in Tool Major

For existing documents, this field displays the current major revision of the document, as defined in the authoring tool, in a read-only format. For new documents, this field is empty.

Current Revision in Tool Minor

For existing documents, this field displays the current minor revision of the document, as defined in the authoring tool, in a read-only format. If the revision scheme does not use minor revision, or if the selected document has not yet been revised, this field is empty.

Revise in Tool Major

From this list box, choose the next available major revision number for the document to revise it locally, without publishing the new information. If you do not want to revise the document at this time, in other words, if you want to reserve revisions numbers without revising the document, leave this field empty.

Revise in Tool Minor

From this list box, choose the next available minor revision number for the document to revise it locally, without publishing the new information. If you do not want to revise the document at this time, in other words, if you want to reserve revisions numbers without revising the document, leave this field empty. If minor revisions are not supported for the document, no options are available in this list.

*IMPORTANT If you do not use the **Minor** field when revising a document for the first time, the minor revision option will never be available for that document for future revisions.

Next Major

Updates the document revision to the next major revision.

SECTION 22

Publishing Documents

When you work in an integrated environment with SmartPlant Enterprise, you must publish documents containing the drawing data and relationships before other authoring tools can share this information. You can publish your documents from the Drawings and Reports task **Management Console** or from a 3D modeling task by using the **Tools > Drawings Console** command.

Before you can publish documents in the software, you must install the SmartPlant Client and the SmartPlant Schema Component, then register the model using the SmartPlant Registration Wizard.

In a 3D task, you can use **SmartPlant > Retrieve** to create and update the Design Basis objects.

The software allows you to publish modified and new objects with the **Changes Only** option. Publish tasks processed through the **Changes Only** workflow are smaller compared to **All** publishes, and are queued to **Load** and **Consolidate** before the merge operation. The merge operation combines the delta data with the previous complete publish data. After the merge operation succeeds, the information is retrievable. The **Changes Only** publishes are not retrievable. To retrieve a **Changes Only** publish, you must perform an **All** publish, which only happens after the load and consolidate processes.

The **Publish** and **Update and Publish** commands are available for the following document types:

- 3D Model Data (SmartPlant Review file type)
 - NOTE The 3D Model Data component is capable of publishing many object types (for example: Piping, Equipment, Cable Trays) depending on the definition of the filter during component setup.
- Orthographic Drawings (viewable file with links to data)
- Isogen Isometric Drawings (viewable file with links to data)
 - NOTE The software supports publishing additional files (for example: PCF, POD) along with the SHA drawing. For more information, see *Enterprise Data* in the *Isometric Drawing Options Reference Guide*. You can access this document using the **Help > Printable Guides** command in the software.
- Reports (viewable file with links to data)

■ NOTES

- The viewable files created when you publish drawings and reports provide relationship links to the 3D Model Data. You must also publish the 3D Model Data to provide the navigation between the viewable files and the 3D Model Data.
- The Publish > Update and Publish command updates and then immediately publishes the selected documents in one step. This command is available only if the model is registered with SmartPlant Foundation.
- For a list of common tasks related to publishing, see *Publish Common Tasks* (on page 426).

- If you are publishing 3D Model Data documents, set the surface style rules and aspects before publishing the documents.
- Every time you generate drawings and reports from Smart 3D in an integrated environment, a SmartPlant Foundation token is used.

When you publish documents, the software:

- Publishes a visual representation of the document that you can view without Smart 3D. For drawings, this is an Intergraph proprietary file, called a RAD file (.sha). For reports, the viewable file is a Microsoft Excel workbook. You can review and mark up the visual representation of the document using SmartPlant Markup Plus or SmartSketch.
- Places the published XML file and any viewable files in the appropriate SmartPlant Foundation vault. This XML file can be retrieved when you are in other authoring tools.

Reasons to Publish

You publish documents and associated data for several reasons:

- Exchanging of data with other tools
- Sharing common data between tools
- Providing enterprise-wide accessibility to published documents
- Managing change, including workflow history, document revisions, and title block information

Revisions and Versions of Published Documents

The first time that you publish a document, the software creates a new document master and the first revision. A revision (major) is an officially recognized change to a document. A version (minor) is an intermediate update that you have published. Revisions can be published for sharing or they can go through an approval process, depending on your needs. Each revisions of a document can have multiple versions.

You can also include revision information within the title block of a drawing by placing drawing property labels within the drawing template.

NOTE When you publish data from any authoring tool, you may not be able to view all of the properties that you published in the SmartPlant Client. You can customize view definitions to allow you to see additional properties. For more information on defining view definitions in the SmartPlant schema, see the SmartPlant Schema Editor User's Guide. For further assistance with viewing data, contact Intergraph Support Services. You can find support information on our web site, http://support.intergraph.com/).

Publishing Title Blocks

The title block is generally displayed at the bottom of a drawing template. It can include signatures, revision and issue information, and other properties associated with the drawing.



You add drawing properties, such as revision information or issue requests, to the title block using the **Place Drawing Property Label** command when editing a template.

When you update the revision information or receive an issue request on a drawing document, the associated properties must be updated. You update the drawing document to incorporate the property changes for any drawing property labels included in the title block.

After you update the drawing document, you can re-publish it and the appropriate title block information is recorded in SmartPlant Foundation. Publishing your drawing document helps you manage the changes, including workflow history, document revisions, and title block information.

The general workflow for including information in the drawing title block is:

- 1. Edit a template and include drawing property labels for revision or issue request *Place a Drawing Property Label on a Template* (on page 393).
- 2. Create drawing documents using the template.
- 3. Publish the drawings Publish documents (on page 431).
- 4. Revise the drawing document or receive an issue request from SmartPlant Foundation *Revise a document* (on page 422) or *Issue request documents* (on page 434).
- Update the drawing documents to include the new revision, issue, or other drawing property information in the title block.

See Also

Delivered Drawing Types (on page 21)
Publishing Documents (on page 424)
Updating Documents (on page 78)

Publish Common Tasks

The following tasks are used to publish documents. If the documents are drawings or reports, the **Publish** command publishes a viewable file with links to the data. If publishing a 3D Model Data document, the software creates a SmartPlant Review file and publishes it.

■ NOTE The viewable files created when you publish drawings and reports provide relationship links to the 3D Model Data. You must also publish the 3D Model Data to provide the navigation between the viewable files and the 3D Model Data.

Setting Properties for Publishing

Before using the Publish command, you should set certain properties on your documents.

If you are publishing 3D Model Data documents, set the surface style rules and aspects as needed before publishing.

Creating and Reserving Revision Numbers

You should create your documents by right-clicking them and selecting **Create Document(s)**. After they are created, if you require revision numbers for the documents, use the **Revise** command to reserve the revision numbers.

Update the Documents

Update the documents, right-click the component and select the appropriate **Update** command. You can also use **Batch > Update** if you are configured to use a batch server.

Publish Data

If you have registered your model using the SmartPlant Registration Wizard, you can publish your documents for retrieval and use in other tools. You can use the **Publish > Update and Publish** command to update drawings and publish them in one step. You can also use the **SmartPlant > Find Documents to Publish** command to generate a list of documents that need to be published and to publish terminations for deleted drawings, isos, and so on.

Publish

Publishes the information in the selected documents. You can access the **Publish Documents** command by right- clicking a component or document.

The **Publish** and **Update and Publish** commands are available for the following types of documents:

- 3D Model Data (SmartPlant Review file type)
- Orthographic Drawings, including Volume and Composed drawings (viewable file with links to data)
- Isogen Isometric Drawings (viewable file with links to data)
- Reports (viewable Microsoft Excel workbook file with links to data)

■ NOTES

- The viewable files created when you publish drawings and reports provide relationship links to the 3D Model Data. You must also publish the 3D Model Data to provide the navigation between the viewable files and the 3D Model Data.
- The Publish > Update and Publish command updates and then immediately publishes the selected documents in one step. This command is available only if the model is registered with SmartPlant Foundation.

★ IMPORTANT When you publish a 3D model, you must now enable the **Scheduler** and **Loader** in SmartPlant Foundation to make the 3D model data document retrievable. The load, consolidate, and merge tasks must complete successfully before the 3D model document can be retrieved.

Define the **Discipline** and **Document Type** properties to enable publishing for the documents. For more information see *Set properties for publishing documents* (on page 428).

■ NOTE For 3D model data, the **Discipline** and **Document Type** properties are already populated.

You may also want to specify documents to be revised, not published, or reserve revision numbers. For more information, see *Revising* (on page 421).

To generate a list of documents that need to be published, you can use **SmartPlant > Find Documents to Publish**. For more information, see *Find Documents to Publish* (on page 441).

You can publish isometric drawings to SmartPlant Foundation in additional file formats other than the **SHA** drawing file format. These additional files along with the **SHA** drawing are generated by the update process. When publishing isometric drawings you can also publish other available data files and reports generated by the update process. For more information, see Published Files (S3D Supplementary).

Publish Dialog Box (on page 435)

Set properties for publishing documents

- 1. In the **Console**, right-click an item in the hierarchy, then select **Properties** on the shortcut menu.
- 2. Go to the WBS Tab.
- Set the **Document Type** property as needed, such as Civil Plan. This property specifies the document subtype when published.
- 4. Set the **Document Style** property as needed, such as Ortho for an orthographic drawing.
- 5. Set the **Discipline** property. If your model has been registered using the SmartPlant Registration Wizard, this property adds the **Publish** command to the right-click menu for the selected document or documents. For a 3D Model Data document, the property is set to **SmartPlant Review Document**. For a drawing or report document, set the discipline to match the type of document.
- 6. Set the Allow Publish property to Yes.

■ NOTES

- If you do not want an item to acquire a property from its parent, select the **Override** column on the **Properties** dialog box, then type a new value. This value propagates to other items deeper in the hierarchy.
- The software treats blank or cleared property values as overrides.
- Before you can publish documents, you must:
 - Install the SmartPlant Client and the SmartPlant Schema Component.
 - Register your model using the SmartPlant Registration Wizard.
 - Revise and update each document.

For more information about configuration, see the *Intergraph Smart*TM 3D *Installation Guide*, available from **Help > Printable Guides**.

Support for Handling Large Publishes

The number of objects published from Smart 3D can become so large that the normal update and publish processing uses up the resources such as memory and address space on the client and server computers. To address these resource issues, Smart 3D now supports the concept of using a cache to keep track of objects that have changed and to only publish these objects. Thus, the software avoids processing objects that have not changed. The mechanisms for doing this are called parallel update, delta publish, and auto-scoping. These processes are not visible to the user.

- Parallel update Updates all the objects. The update process determines the number of
 processors on the system and uses this information to partition the objects so that multiple
 updates can happen at the same time. Because the updates are happening in parallel, the
 system can update a larger number of objects faster.
- Delta publish Leverages the cached information provided by parallel update to determine which objects must be published. Because only changes are published, you can produce larger files in the data warehouse.

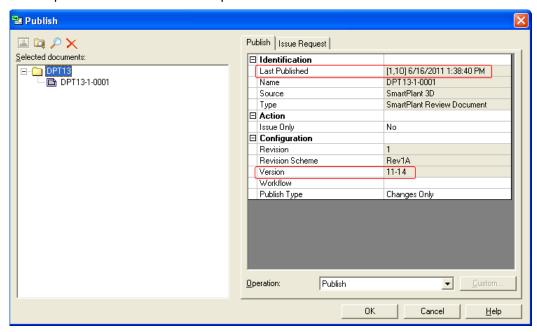
Because the complete set of objects will not be published each time, it is up to Smart 3D to keep track of deleted and moved objects. These moved or deleted objects will be sent with

the published objects, eliminating the need to probe the adapter for the existence of missing objects.

IMPORTANT When you publish a 3D model, you must now enable the **Scheduler** and **Loader** in SmartPlant Foundation to make the 3D model data document retrievable. The load, consolidate, and merge tasks must complete successfully before the 3D model document can be retrieved.

Both parallel update and delta publish can publish larger documents by automatically splitting the data into smaller sections in Smart 3D. If your publish data is split into smaller sections, the document version number increments for each section.

For example, if you publish a document that is split into four smaller sections, the **Version** field contains all resulting version numbers. In the following dialog box, the published documents are published to SmartPlant Foundation as 4 different versions (11-14). The last version contains all of the updated information from the publish.



Published Documents - Load, Consolidated, and MergeDelta Tasks (S3D)

A tool published document results in a document version object in SmartPlant Foundation. A document version has four files attached to it.

- 1. **Data XML file** Contains published objects, relationships and correlations (SameAs relationships).
- 2. **Meta data XML file** Contains meta data about the published document version.
- Instructions XML file Contains instructions for deleted objects and relationships. Also
 contains resurrect instructions for objects that were deleted previously and resurrected as a
 result of restoring a tool database.
- 4. **View file** This is a file in the tool format that represents the (graphical) view of the published document.

When a document is submitted to a workflow that has a load process step, the following tasks are created.

- NOTE If the document is not published to such a workflow, and when the **Load document** command in SmartPlant Foundation Desktop Client is run on the published document, the same tasks are created.
- Load Task This task processes the objects and relationships in the data XML by loading them into a publish domain. This task also processes the instructions in an Instructions XML file. For example, the delete instruction would result in termination of the object referenced in the delete instruction.
- Consolidate Task This task processes the correlations (SameAs relationships) in the data XML file. A SameAs is a correlation relationship published by tools indicating that a published object is identical to an object published by another tool. The correlation of the object in the current publish domain (say, local object) to the external object (object published by another tool in another publish domain) is done by creating a shared object in the Data Warehouse domain. The local and external objects are linked through SPFComprisedOf relationships; for example, one SPFComprisedOf relationship between the local object and the shared object, and a second SPFComprisedOf relationship between the external object and the shared object.
 - NOTE An object that is not correlated also has a shared object created in the Data Warehouse domain in preparation for future correlation.

The consolidate task also creates a hidden file, known as graphical mapping file, which has mappings between objects in the database and graphical elements in the view file. This file is used by SmartPlant Foundation Desktop Client for navigation between published objects in the list view and their graphical representations in the view file.

MergeDelta Task - This is a task created for delta publishes of Smart 3D. A delta publish contains new, modified and deleted objects in a document since the last publish of that document. Only Smart 3D has the capability to do a delta publish. Other tools publish full data where every object is published whether it is modified or not since last publish.

Because a delta published data XML file and view file only contains the delta, it is required that it becomes a full data XML and full view file for complete view of the data and graphics of that document. The MergeDelta task converts the delta data XML file into a full XML file by merging data from the previous full XML file into the delta XML file.

- ★ IMPORTANT When you publish a 3D model, you must enable the Scheduler and Loader to make the 3D model data document retrievable. The load, consolidate, and merge tasks must complete successfully before the 3D model document can be retrieved. For more information, see Schedulers and Queue Management.
- ▶ NOTE If you are using SmartPlant Basic Integrator, only the merge task must complete successfully before the 3D model document can be retrieved. For more information, see Schedulers and Queue Management.

The merge involves the following:

- a. Replace modified objects and relationships in the full publish XML, view file, and the hidden graphic map file.
- b. Delete objects and relationships that are identified in the delete, and move instructions in the delta instructions XML file, view file, and the hidden graphic map file.
- Add new objects and relationships to the full publish XML, view file, hidden graphic map file.

d. Replaces the delta XML file in the vault with the updated full publish XML file, view file, and the hidden graphic map file.

Set SmartPlant Foundation to Tolerate Failed Loads

For working in any integrated environment, a new site configuration flag in SmartPlant Foundation Server Manager allows the SmartPlant Foundation Loader to skip a published document that failed during load and continue processing other documents in the queue.

In SmartPlant Foundation Server Manager, set the **SkipFailedDocumentsInLoad** flag to **True** to allow failed documents to be skipped and other documents to continue processing. Typical behavior is that after a document is successfully loaded, any previous failed document is removed from the queue.

The behavior during a Smart 3D publish is different. If any Smart 3D publish, either a full publish or delta publish, fails to load, the failed versions remain in the load queue until a full publish successfully loads. If a delta publish loads successfully following these failures, it does not remove any previous failed version from the queue because the data in delta publishes is different from version to version, so a failed previous version needs to stay in the queue until you resolve the failure or a full publish is loaded. A second reason is this gives an indication to you that a full publish is needed. When a full publish is loaded, all failed delta publishes are removed from the queue.

Publish documents

Before you can publish documents, you must ensure that your computer is configured properly. The configuration includes installing the SmartPlant Client and the SmartPlant Schema Component and registering the model through the SmartPlant Registration Wizard.

You must use the **SmartPlant > Retrieve** command in one of the 3D tasks to import published data.

- ★ IMPORTANT When you publish a 3D model, you must now enable the **Scheduler** and **Loader** in SmartPlant Foundation to make the 3D model data document retrievable. The load, consolidate, and merge tasks must complete successfully before the 3D model document can be retrieved.
- NOTE The SmartPlant menu is not available in all tasks.
- 1. Right-click a component and select **Publish**. The **Publish** dialog box displays.

■ NOTES

- If the Publish command is not available on the shortcut menu for the component or document, check the document properties and make sure that the documents are up-to-date and have been revised first.
- You can use the Publish > Update and Publish command to update drawings and publish them in one step. This command is available only if the model is registered with SmartPlant Foundation.
- 2. Edit information as necessary for the selected documents.
 - When multiple documents are selected, only property values shared by all of the selected documents display in the table. Changing a value in the table changes that value for all of the selected documents.
- 3. Select the **Publish Type**.

Smart 3D supports Publish Type for delta publish.

■ NOTES

- The default setting for Publish Type is Changes Only. If the 3D model has not been published before, the software automatically selects All and performs a complete publish to ensure all filtered data is published and loaded into SmartPlant Foundation.
- For subsequent publishes, select All to publish all filtered data in the 3D model, even if it is unchanged. Select Changes Only to publish only the changes in the 3D model since the last successful publish.
- If the 3D model does not publish successfully during a delta publish, select All so that data from the previous publish is published and loaded into SmartPlant Foundation.
- Other SmartPlant applications do not support **Publish Type**. For these SmartPlant applications, the default setting for **Publish Type** is **All**, and it cannot be changed.
- 4. Select one of the following operations:
 - Publish to publish the selected documents immediately.
 - Background publish to publish as a separate process so that you can continue working in the application.
 - Scheduled publish to publish in the batch mode by the authoring tool. The documents
 are not published immediately. Instead, the selected documents are scheduled for
 publish at a later time and may be scheduled as a recurring operation.
- 5. Click **OK** to publish the selected documents.

■ NOTES

- You can verify the publishing process by starting the SmartPlant Client on your computer and searching for the published document.
- When publish is complete, the following message displays: Documents have been published successfully. If the View Log button is enabled, messages are available concerning the operation. These messages include errors, warning, and informational messages. Click View Log to review these messages.

Publish Workflows

When a SmartPlant application publishes, the user can publish documents using a workflow. The following list describes the delivered publish workflows and use cases for the workflows.

Workflow	Workflow Description	Workflow Use Cases
Auto-Correlate	Compares the objects in a published document to the objects that are already in SmartPlant Foundation. If the object correlates (by name) to an object with the same shared object definition, SmartPlant Foundation automatically correlates the objects. If the object can correlate to more	 When a publishing tool does not have correlation functionality. When a publishing tool does not retrieve. When using SmartPlant Foundation to perform consistency checking and the

Workflow	Workflow Description	Workflow Use Cases	
	than one object through multiple shared object definitions, a task is placed in the To Do list to allow the user to select the correlation.	authoring applications do not retrieve. When Shared Objects To Correlate option is used to configure which types of objects are auto-correlated.	
AutoLoadPublish	Generates a document load task and consolidate task for the publishing document. These tasks are performed by the SmartPlant Foundation Scheduler.	Use AutoLoadPublish when you intend to load the published document data into SmartPlant Foundation.	
AutoLoadPublish Merge	Publish large 3D models. Creates the published 3D document, loads the 3D data in SmartPlant Foundation, then merges these object sets into a new composite document that represents the entire 3D model.	Use AutoLoadPublishMerge when you intend to view or navigate multiple 3D models as one composite 3D model document in one 3D view.	
Correlate	Places published objects with possible correlations in the To Do list. From the To Do list, the user manually correlates selected items. SmartPlant Foundation creates a shared object.	Correlate works similar to Auto-Correlate except you can view the possible correlations before the actual correlation takes place.	
Publish	Takes the document through an approval step. Upon approval, it creates the document load and consolidate tasks.	Use Publish when you want the published document approved before it is loaded into SmartPlant Foundation. The approval step is added to the To Do list.	
PublishWithDocL oad	Loads document into a document configuration and submits document to approval step. Upon approval, it creates document the load and consolidate tasks.	Use PublishWithDocLoad when you want to view the document data in SmartPlant Foundation before approval. The approval step is added to the To Do list.	
PublishForLoadO nly	Generates a document load task and consolidate task for the publishing document and makes the documents irretrievable by any tool. These tasks are performed by the SmartPlant Foundation Scheduler.	Use PublishForLoadOnly when you want to publish documents to view, navigate and also for consistency checking in SmartPlant Foundation without the possibility of the document being retrieved by any tool.	

Publish a large 3D model to SmartPlant Foundation

Publishing large 3D models to SmartPlant Foundation has reached memory and file transfer limitations. To make publishing 3D models more efficient, you can publish models in distinct object sets that accommodate your memory resources. Distinct object sets mean that an object in the 3D model must appear in only one published model. You publish these object sets using a workflow called **AutoLoadPublishWithMerge**. This workflow has **MergePublishedFile** process step configured. This workflow creates the published 3D document, loads the 3D data in SmartPlant Foundation, then merges these object sets into a new composite document that represents the entire 3D model. The name of the composite document is same as the name of the workflow to which the Published 3D model is submitted. If you have existing published 3D models in SmartPlant Foundation and want to merge these into one representative model document, you can create a 3D composite document manually, and then relate the published models to the document.

- 1. Create filters for segregating and selecting data that will be in the published model documents. For example, you can create filters for equipment, piping, structural, and HVAC.
- 2. In the **Drawings and Reports** task, create your separate model documents.
- 3. Publish each model document separately.
 - NOTE The software publishes a new version of the 3D document, for example, a model document called Equipment-1-001, to SmartPlant Foundation. This document is related to AutoLoadPublishWithMerge, a zvf file, and a mapping file.
- On the Publish tab on the Publish dialog box, select the AutoLoadPublishWithMerge from the Workflow list.

Issue request documents

Before you can publish documents, you must ensure that your computer is configured properly. The configuration includes installing the SmartPlant Client and the SmartPlant Schema Component and registering the model with the SmartPlant Registration Wizard. For more information, see the *Intergraph Smart*TM 3D *Installation Guide* available from **Help > Printable Guides**. For more information on the steps involving the SmartPlant Foundation Desktop Client, see the *SmartPlant Foundation Desktop Client User's Guide*.

NOTE This functionality is only available in project mode. It is not supported in integration mode.

You must use the **SmartPlant > Retrieve** command in the Common task to import published data.

- 1. Right-click a component and select **Publish**. The **Publish** dialog box displays.
 - NOTE If the **Publish** command is not available on the shortcut menu for the component or document, check the document properties and make sure that the documents are up-to-date. For more information, see *Set properties for publishing documents* (on page 428).
- 2. Click the Issue Request tab.
- 3. In the **Issue to** field, select the contract that you want to assign the document or documents.
- Under Selected documents, select the documents that you want to associate with the specified contract.

- 5. Click Add to add the documents to the Issue Request list. To remove documents from the list, select them and click Remove. Click Engineering Tools to add documents from engineering tools, such as P&IDs or PFDs. Click File System to add documents from another file system, such as Microsoft Word documents or Microsoft Excel workbooks.
- 6. Click **OK** to issue the contract request for the selected documents.
- Start SmartPlant Foundation Desktop Client on your computer and search for the published document to verify the publishing process.
- 8. Right-click the document in the Desktop Client tree view and select Refresh.
- 9. Return to the Drawings and Reports task and update the document to incorporate the new Issue information. For more information, see *Updating Documents* (on page 78).
- 10. Review the Issue properties. Right-click the document and select **Properties**. Select the **Issue** tab to see the Issue information. You can also open the document to see the Issue information in the title block if you added it.
- 11. Publish the document with the updated Issue information. For more information, see *Publishing Documents* (on page 424).

■ NOTES

- Only updated documents can be published.
- You can verify the publishing process by starting the SmartPlant Client on your computer and searching for the published document.

See Also

Find Documents to Publish (on page 441) Publish (on page 427) Publish documents (on page 431) Publishing Title Blocks (on page 425)

Publish Dialog Box

Provides a list of documents selected to publish.

Publish Tab (Publish Dialog Box) (on page 435) Issue Request Tab (Publish Dialog Box) (on page 439)

See Also

Publish documents (on page 431)
Find Documents to Publish (on page 441)
Publish (on page 427)
Document Properties Dialog Box (on page 440)

Publish Tab (Publish Dialog Box)

Displays the properties of the selected document or documents. If only one document is selected in the tree view, the properties displayed on this tab are the properties of that one document. If multiple documents are selected, only the properties with the same value for all documents display. Any properties with varying values across the documents display with blank values in these fields.

You can change some of the values assigned to one or more documents by changing the value displayed in the table. The value you type here overrides any existing values for all selected documents.

Selected documents

Displays a list of the documents selected for publishing. You must populate this list by selecting documents in the **Management Console** or **Detail View** before you use the **Publish** command. For each document, this list displays the name, the type of document, the workflow from which the document was last published, the revision and version numbers, the revision scheme, and the date when the document was last published.

Engineering Tool

Opens a dialog box to select documents to add to the **Selected documents** list. This functionality is not available in the current release.

File System

Opens a standard Microsoft dialog box that allows you to select documents to add to the **Selected documents** list. When you select a file with this **Select File** dialog box, the **Document Properties** dialog box displays, allowing you to specify information about the file, such as whether it is a new file; the category, type, and subtype of the document; and the name, description, and title of the document.

Find

Opens the **Find Documents to Publish** dialog box, which allows you to search for documents to add to the **Selected documents** list. For more information, see *Find Documents to Publish Dialog Box* (on page 442).

Last Published

Indicates the date on which the document or documents were last published.

Name

Displays the name of the document.

Source

Indicates the authoring tool in which the document was created.

Type

Displays the type of document or documents selected.

Comment

Allows you to type information about the selected documents that are publishable.

Issue Only

Allows you to issue request documents without, necessarily, republishing them. Use this option when no changes were made to a drawing and you only want to add it to a contract.

TIPS

- Even with this option set, you can still publish the documents. If any of the documents have never been published, they must be published, regardless of this setting.
- You will receive an error message if you select multiple documents and activate this option when one or more of the selected documents cannot be changed. For example, the error message displays if the selected set of documents includes both a new document (for which this field can be set only to No) and current or locked documents (for which this field can be set only to Yes). The error message prompts you to select a smaller set of documents.

Owning Group

Select an owning group from the drop down list to which the document belongs.

■ NOTES

- By default, the owning group selected for the previous version, if any, is shown.
- All the owning groups configured in SmartPlant Foundation are listed.

Revision

Displays the current revision number of the selected document or documents.

*TIP You will receive an error message if you attempt to change the value in this field when you have selected one or more documents that have conflicting revision schemes or different possible revisions. The error message prompts you to select a smaller set of documents.

Revision Scheme

Displays the revision scheme applied to the selected document or documents.

NOTE Only revision schemes that are applicable to the configuration (plant) or classification (document type) are available in the shortcut menu. The revision schemes related to a configuration or classification are not available for any other configurations or classifications. If none of the revision schemes are related to the configuration or classification, then all revision schemes are available unless they are related to any other configuration or classification. For more information on revision scheme configuration, see *Configuring Different Revision Scheme Strategies* in the *How to Configure Document Management* guide.

TIP You will receive an error message indicating that this field cannot be edited if one or more of the documents that you have selected are not new or will have a revision scheme supplied by the authoring tool. The error message prompts you to select a smaller set of documents.

Version

Indicates the current version of the document or documents.

Workflow

Indicates the workflow to which the selected document or documents are assigned.

Publish Type

Allows you to publish all data in a 3D model or only the changes to the model since the last successful publish. If the 3D model has not been published before, the software automatically selects **All** and performs a complete publish to ensure all filtered data is published and loaded into SmartPlant Foundation.

Publishing documents	Publish Type	
	All	Changes Only
First publish after creating the document	All objects	All objects
Subsequent publish with no changes to the model	All objects	No objects

Publishing documents	Publish Type	
Subsequent publish with changes to the model	All objects	Changed objects
Subsequent publish with no changes to the model (after changing the Publish Type)	All objects	No objects
Subsequent publish with changes to the model (after changing the Publish Type)	All objects	Changed objects

TIP You will receive an error message indicating that this field cannot be edited if one or more of the documents that you have selected have conflicting sets of possible workflows. The error message prompts you to select a smaller set of documents.

Check and publish released claims for previously deleted items

Specifies that you want to resolve issues where deleted items were restored from an earlier version and the claim on them was released. This check takes additional time and should only be used when deleted items have been restored. This option is not supported in this release.

TIP This check box should also be activated when publishing after a backup is restored or when releasing the claim on an object forces another tool to release the claim on a related object that was previously deleted. In this specific case, the tool fetches the object from As-Built again and releases the claim.

Operation

Specifies the operation to perform on the selected documents.

- Publish Selected documents are published immediately.
- Background publish Selected documents are published immediately as a separate process, allowing you to perform other tasks at the same time.
- Scheduled publish Selected documents are published in the batch mode by the authoring tool. This option is available only for tools that support batch mode and are processed by the authoring tool, not the SmartPlant Client. The documents are not published immediately. Instead, the selected documents are scheduled for publish at a later time and may be scheduled as a recurring operation.

NOTE If the software cannot make a SmartPlant Foundation server connection when you use **Scheduled Publish**, you are prompted to provide a valid SmartPlant Foundation login and password.

Custom

Opens the **Custom** dialog box. This functionality is available only if defined by your project implementation team.

Check for deleted objects no longer on documents

Select **Check for deleted objects no longer on documents** option if you want to process the move instructions while publishing.

See Also

Publish documents (on page 431)
Find Documents to Publish (on page 441)

Publish (on page 427)
Document Properties Dialog Box (on page 440)

Issue Request Tab (Publish Dialog Box)

Displays the documents associated with a specific issue request and allows you to add documents to or remove documents from a request.

Selected documents

Displays a list of the documents selected for publishing. You must populate this list by selecting documents in the **Management Console** or **Detail View** before you use the **Publish** command. For each document, this list displays the name, the type of document, the workflow from which the document was last published, the revision and version numbers, the revision scheme, and the date when the document was last published.

Engineering Tool

Opens a dialog box to select documents to add to the **Selected documents** list. This option is not supported in this release.

File System

Opens a standard Microsoft dialog box that allows you to select documents to add to the **Selected documents** list. When you select a file with this **Select File** dialog box, the **Document Properties** dialog box displays, allowing you to specify information about the file, such as whether it is a new file; the category, type, and subtype of the document; and the name, description, and title of the document.

Find

Opens the **Find Documents to Publish** dialog box, which allows you to search for documents to add to the **Selected documents** list.

Issue to

Contains a list of all objects (contracts) that can support issue requests. When you select an item from this list, the names of any documents associated with that object display in the table.

Add

Creates a new item in the table for any documents highlighted in the **Selected documents** tree view.

Remove

Deletes a selected document from the table.

Document Name

Displays the names of all documents associated with the object in the **Issue to** field.

See Also

Publish documents (on page 431)
Find Documents to Publish (on page 441)
Publish (on page 427)
Document Properties Dialog Box (on page 440)

Document Properties Dialog Box

Provides details about a new or existing document selected for publishing. You can access this dialog box using the following procedure:

- 1. Click **File System** on the *Publish Dialog Box* (on page 435). A Microsoft standard **Select File** dialog box displays.
- 2. Select a file to display the **Document Properties** dialog box and specify information about the file.
- TIP An asterisk (*) next to a field indicates that the field must be completed before the software enables the **OK** button.

Selected file

Displays the name of the file that you selected on the **Select File** dialog box.

New document

Indicates that this document has not previously been published.

Published previously

Indicates that the file has already been published at least once before.

Document category

Select the category for the document.

Document type

Select the type of document. The options that display in this list are determined by the selection that you make in the **Document category** field.

Document subtype

If applicable, select the subtype for the document. The options that display in this list are determined by the selection that you make in the **Document type** field.

Name

Type the name of the file as it will be known in the integrated environment.

Descriptions

Type a brief description of the file. This description displays later to help you recognize the file.

Title

Type the official title of the document.

See Also

Find Documents to Publish Dialog Box (on page 442) Publish Dialog Box (on page 435) Issue Request Tab (Publish Dialog Box) (on page 439) Publish Tab (Publish Dialog Box) (on page 435)

Find Documents to Publish

Generates a list of documents that either have not been published or have been modified and need to be published again or have been deleted since the last publish. The command is found on the **SmartPlant** menu in the Drawings and Reports task.

TIP This command can also be accessed from using the **Find** button on the **Publish** tab of the **Publish** dialog box. For more information, see *Publish Tab* (*Publish Dialog Box*) (on page 435).

Documents must be up-to-date, and the required **Discipline** property must be defined in order for the documents to be available for publishing. An error message displays if one or more of the documents found by the **Find Documents to Publish** command do not meet this criterion. All items matching the publish criteria continue through the process. For more information on setting the appropriate properties, see *Set properties for publishing documents* (on page 428). For more information on updating documents, see *Updating Documents* (on page 78).

The **SmartPlant > Find Documents to Publish** command looks for the following:

- Documents created but never published
- Documents modified since their last publish
- Documents deleted after being published

The command looks for documents that need to be republished in the active WBS project. For example, if **Project A** is the active project, the **Find Documents to Publish** command looks for documents in **Project A** only. You set the active WBS project in the **Active Project** box on the main toolbar. For more information, see *Manage Projects* (on page 442).

Find Documents to Publish Dialog Box (on page 442)

Find documents to publish

Documents must be up-to-date, and the required **Discipline** property must be defined in order for the documents to be available for publishing. An error message displays if one or more of the documents found by the **Find Documents to Publish** command do not meet this criterion. For more information on setting the appropriate properties, see *Set properties for publishing documents* (on page 428). For more information on updating documents, see *Updating Documents* (on page 78).

1. From the Drawings and Reports task, click **SmartPlant > Find Documents to Publish**.

TIPS

- This command is available only if you have registered the model using the SmartPlant Registration Wizard.
- This feature is also available by clicking the **Find** button on the **Publish** dialog box.
- The Find Documents to Publish command determines which documents need to be published or re-published and displays the results of the search in the Find Documents to Publish dialog box.
- 2. From the **Select documents to publish** list, check the boxes corresponding to the documents that you want to publish.
 - TIP You can quickly select the entire list by clicking **Select All**, or you can clear the entire list by clicking **Clear All**.

 Click OK to accept the selections. The documents selected for publishing now display in the Documents to Publish list on the Publish dialog box and can be saved by publishing the documents. For more information, see Publish documents (on page 431).

Find Documents to Publish Dialog Box

Allows you to search for documents that have been updated since they were last published. Additionally, you can use this dialog box to terminate documents that were previously published but no longer exist in the authoring tool. You can access the **Find Documents to Publish** dialog in two ways:

- Select SmartPlant > Find Documents to Publish.
- Click Find on the Publish command dialog box.

Last Published

Displays the date when the files were last searched. The information displaying in the lists on this dialog box was found on this specified date and time. This option is not available in the current release.

Update

Displays the Update dialog box, which allows you to define new search criteria for finding documents to publish. This option is not available in the current release.

Select documents to publish

Displays a list of files that were either updated since they were last published or files that have not yet been published. For each file, this list displays the file name and type, and the date on which the document was last published. If the file has not been published, the **Last Published** field for the document is **New**.

Select documents to terminate

Displays a list of the files that were previously published but have since been removed from the project. For each file, this list box displays the file name and type, and the date on which the document was last published.

Select All

Selects all of the files in the associated list of documents.

Clear All

Clears any selected documents in the associated list.

See Also

Publishing Documents (on page 424)
Find Documents to Publish (on page 441)
Publish Tab (Publish Dialog Box) (on page 435)

Manage Projects

The Work Breakdown Structure (WBS) project is shown in the dropdown at the upper left-hand corner of the Drawings and Reports task window, next to the **Permission Group** dropdown. It shows the current active project.



In the Common task, you can create new WBS items and projects or edit existing ones. For more information, see the *Common User's Guide*.

You use projects in conjunction with publishing. The active project must be set before using the **Final Publish** command on the **SmartPlant** menu.

See Also

Publishing Documents (on page 424) Select Active Project Dialog Box (on page 443)

Select Active Project Dialog Box

Specifies the active project. You can access the **Select Active Project** dialog box by clicking **More** in the **Active Project** box on the main toolbar. You define whether you want to look in the local **Workspace** or in the **Database** for the project by selecting the options at the top of the dialog box. The project hierarchy updates with the selection of the option.

See Also

Manage Projects (on page 442)

APPENDIX A

Appendix: Troubleshooting Drawings and Reports

This section describes Drawings and Reports error levels and error logging. You can use log files to review activities and errors that occur when working with the software.

Error Levels

Run the [Product Folder]\Core\Tools\Administrator\Bin\EnableErrorLog.exe tool to enable detailed error logging in the SP3DErrors.log. For more information, contact Intergraph Support Services. You can find support information on our web site at http://support.intergraph.com\.

Generally speaking, the larger the number for the error level, the more exhaustive is the logging of errors. The error levels are as follows:

- 1 General user error. This is the default level. At this level, log files only contain error messages for certain anticipated error conditions (such as a missing filter or a missing view style), as well as some unanticipated error messages.
- 101 Development-specific error level. At this level, log files include everything from the previous error level, as well as certain development-specific error or informational messages.
- 201 General Information. At this level, log files include everything from the previous error levels, as well as informational messages about what projects and methods are being called.
- 251 Batch Information. At this level, log files include everything from the previous error levels, as well as special batch-specific informational messages.
- 999 Exhaustive. At this level, log files contain all informational and error messages. When the error level is set to 999, the error log files can become very large.

Log Files

There are three categories of orthographic drawings error log files. These files are all located in your temporary (Temp) folder. For example, your Temp folder might be located at C:\Documents and Settings\login name\Local Settings\Temp.

TIP You can type **%Temp%** in the **Address** box at the top of Windows Explorer to go to your Temp location.

The log files are:

- %TEMP%\Drawings.log and %TEMP\Errors.log General purpose error log file. Most of the errors encountered in Drawings and Reports are logged here.
- C:SP3DBatchSvcTemp\Drawings_Batch.log The batch service-specific error log file. Any errors or information messages related to the batch service are found here.
- C:SP3DBatchSvcTemp\DwgBatchServer_< Process ID >_< yyyy_mm_dd_hh_nn_ss >.log The batch server-specific error log files. Any errors or information messages related to the
 batch servers are found here.

- C:SP3DBatchSvcTemp\DwgBatchTier_< Process ID >_< yyyy_mm_dd_hh_nn_ss >.log The batch tier-specific error log files. Any errors or information messages related to the
 batch tier services are found here.
- Generation-time error logs (for example, error log files generated during a drawing update operation). You can easily view these files by right-clicking a drawing in the software and selecting View Log on the shortcut menu.

TIP The Temp location also includes .xml files. The software creates one .xml file for each graphical view in a drawing as it processes the drawing.

Out of Memory Tips

If you are receiving Out of Memory errors when processing very large drawings, check the following:

- Set the Intersection Edges option in the view style to Off. Sometimes, this option is set to High Resolution for large drawings. High Resolution is typically useful for drawings of one object, such as a hanger drawing or an equipment drawing, where you want to see detail such as the coped intersection of a nozzle with a tank cylinder. If you have this option set high in a large plan drawing, then the software examines every small beam, cylinder, or nut and tries to draw coped intersections or rounded edges in the webs. This operation uses a lot of time and memory.
- Make sure the Preserve Z Order option in the view style is turned off unless you really need it. A case where you might need it is in a cable tray drawing where trays are stacked or crossing in plan. However, for most drawings, this setting just leads to increased processing time and high memory usage for little gain.
- Make sure multiple aspects have not been chosen in the VHL graphic rules.
- If you have a very large session active in the host and then update a drawing, the update process starts at the large memory usage in the active session. So, if you limit your workspace to a very small set of objects or even one object, the update process has a better chance of succeeding. If you use batch updates instead of local updates, workspace is not an issue unless you are using your local computer as the batch server.
- Decrease the **Flush Threshold** setting (the default is 2000). This setting is available on the **Drawing View Properties** dialog box for a view in a composed drawing. In some instances, increasing the number may help. During the update process, the Drawings software asks Core for the monikers of all the objects it has to process. Core passes in the monikers and Drawings binds them, thus using memory. The software binds up to 2000 of the objects before it releases the memory and grabs the next 2000. If you reduce this number, say in increments of 500, the memory gets released much more often and can lead to a successful update. It is an iterative process to figure out a good number. This value should be from 5 to 5000 with a default of 2000.
- Set the Geometry Validation setting to ON (the default is OFF). This setting is available on the Drawing View Properties dialog box for a view in a composed drawing or a marine mode drawings-by-rule drawing. If the drawing has SAT or DGN files, there are sometimes many invalid geometry errors that can use increasing amounts of memory. With this setting as On, the software analyzes geometry, then discards and does not draw invalid geometry, thus freeing up resources and reducing update time. The caveat is that the drawing may not be 100% accurate because invalid geometries may represent legitimate items but are removed from the drawing. If the reference file is there just as background, this situation might be acceptable.

- In general, you should not place huge SAT files as equipment shapes. If you must place them as shapes, break them into smaller files.
- The range inside reference files should be kept as small as possible. For example, if the file contains a pipe rack far away from the global origin, place the file in the model and then move it to the appropriate position.

Isometric Drawings

The isometric drawing log files reside at the location specified in your Temp environment variable. For example, the path to the log might be C:\Documents and Settings\login name\Local Settings\Temp.

You can view message files (.mes), piping component files (.pcf), and .xml files for the isometric drawing generation process.

VHL Precision

Objects can be displayed as Vector Hidden Line (VHL). There can be instances when you may want the hidden line information for a model. For example, it may be useful to know how the edges in the hidden line view are occluded. In general, **VHL Precision** should not be set, unless you have some parts in the model that have been modeled with a precision different from rest of the system. The values you can set impact the VHL calculations. The **VHL Precision** setting must be a positive number between 0.001 and 0.000001. This setting is available on the **Drawing View Properties** dialog box for a view in a composed drawing or a marine mode drawings-by-rule drawing.

ISOKEEPFILES Variable

This variable exports XML settings to the location specified in your Temp environment variable. An XML file contains the name of the isometric view style currently being used. This information can be used to troubleshoot isometric drawing settings.

TIP The XML file containing the isometric view style will be approximately 27 KB in size.

The ISOKEEPFILES variable is not a Drawings and Reports setting, it is a System Properties setting.

- 1. Click **Start** and right-click **My Computer**.
- Select Properties.
- 3. Select the Advanced tab.
- 4. Click Environment Variables.

The Temp environment variable is listed under **User variables for** *username*. If you are unsure of where your Temp folder is, the location is noted in this box.

- 5. On the Environment Variables dialogue box, select New under the System variables box.
- 6. Type ISOKEEPFILES in the Variable box and type YES in the Value box.
- 7. Click OK.

NOTE You must set this variable before entering Drawings and Reports and creating the isometric drawings.

Reports

The log file for reports (SP3DReports.log) resides at the location specified in your Temp environment variable. For example, the path to the log might be C:\Documents and Settings\login name\Local Settings\Temp.

See Also

Drawings and Reports (on page 16)
Troubleshooting Linked Servers (on page 447)

Troubleshooting Linked Servers

If your Site/Catalog/Model database server is different from your Reports database server, you can use linked servers for communication between the data sources. However, if linked servers are not configured correctly, the login may fail when you run queries against a linked server.

For linked servers to work correctly, the following must be true:

- The database link must be created on the Site/Catalog/Model database server, not the Reports database server.
- The linked server must support **SQL Server and Windows Authentication Mode**.
- You must be connected to SQL Server using Windows Authentication Mode on both servers.
- Security account delegation must be available on the client and the sending server.

For more information about setting up linked servers, see Microsoft SQL Server documentation.

See Also

Appendix: Troubleshooting Drawings and Reports (on page 444)

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